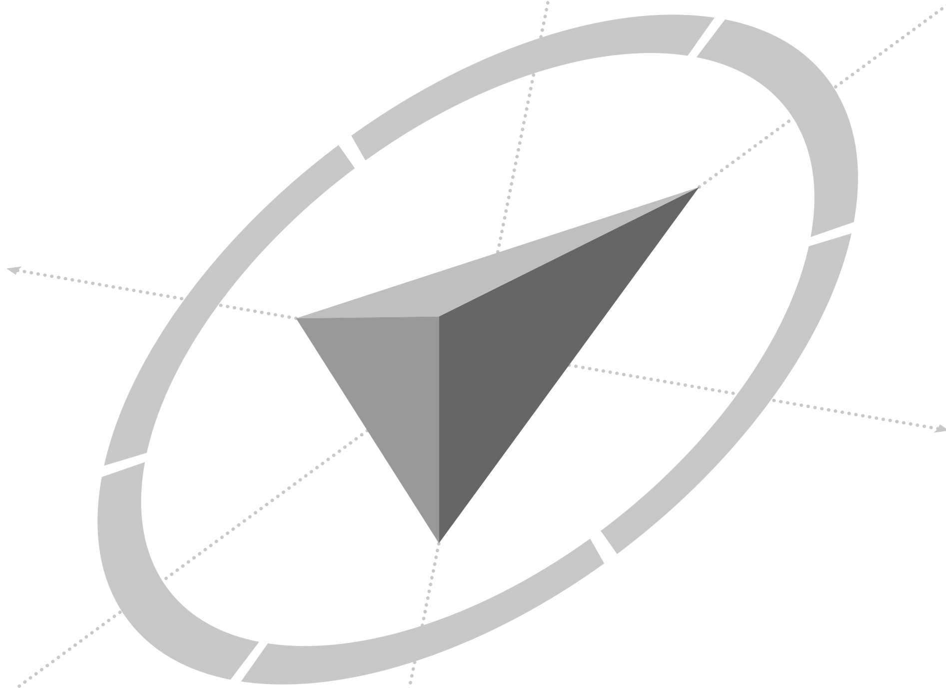
PIPESYS

Version 1.13



User's Guide

Documentation Credits

Authors of the current release, listed in order of historical start on project:

Rolf C. Fox, B.Sc; Edmard A. DeSouza, B. Math; Garry A. Gregory, Ph.D., P.Eng.; Lisa Hugo, BSc, BA; Chris Strashok, BSc

Since software is always a work in progress, any version, while representing a milestone, is nevertheless but a point in a continuum. Those individuals whose contributions created the foundation upon which this work is built have not been forgotten. The current authors would like to thank the previous contributors.

A special thanks is also extended by the authors to everyone who contributed through countless hours of proof-reading and testing.

Contacting Hyprotech

Hyprotech can be conveniently accessed via the following:

Website:	www.hyprotech.com
Technical Support:	support@hyprotech.com
Information and Sales:	Info@hyprotech.com

Detailed information on accessing Hyprotech Technical Support can be found in the **Technical Support** section in the preface to this manual.

Table of Contents

1 Overview 1-1

1.1 Introduction.. 1-3
1.2 How This Manual Is Organized 1-5
1.3 Disclaimer.. 1-5
1.4 Copyright... 1-6
1.5 Acknowledgements 1-6
1.6 Warranty.. 1-7
1.7 Technical Support 1-8

2 Installation 2-1

2.1 System Requirements... 2-3
2.2 Software Requirements... 2-3
2.3 Installing PIPESYS.. 2-4

3 The PIPESYS View 3-1

3.1 PIPESYS Features.. 3-3
3.2 Adding PIPESYS.. 3-4
3.3 PIPESYS User Interface 3-8
3.4 The Main PIPESYS View 3-9

4 Elevation Profile -Quick Start 4-1

4.1 Flow Sheet Set-Up 4-3
4.2 Adding the PIPESYS Extension.. 4-4
4.3 Defining the Elevation Profile 4-5

5 Pipe Unit View 5-1

5.1 Connections Tab 5-3
5.2 Adding a Pipe Unit.. 5-9

6 Global Change Feature.. 6-1

6.1 Global Change View.. 6-4
6.2 Global Change Procedure... 6-7
6.3 Making a Global Change.. 6-9

7 In-line Compressor 7-1

7.1 The Compressor View 7-3

7.2 Adding a Compressor 7-13

8 In-line Pump 8-1

8.1 In-line Pump View 8-3

9 In-line Facility Options 9-1

9.1 In-line Heater 9-3

9.2 In-line Cooler 9-4

9.3 Unit-X 9-6

9.4 In-line Regulator 9-8

9.5 In-line Fittings 9-9

9.6 Pigging Slug Check 9-11

9.7 Severe Slugging Check 9-13

9.8 Erosion Velocity Check 9-16

9.9 Side Stream 9-18

10 Gas-Condensate Tutorial 10-1

10.1 Setting Up the Flowsheet 10-3

10.2 Adding a PIPESYS Extension 10-8

10.3 Applying a Global Change 10-16

11 PIPESYS Application 1 11-1

11.1 Gas Condensate Gathering System 11-3

11.2 Setting up the Flowsheet 11-6

11.3 Setting Up the Case 11-8

11.4 Results 11-18

12 PIPESYS Application 2 12-1

12.1 Optimization Application 12-3

13 Glossary of Terms 13-1

13.1 PIPESYS Terms 13-3

13.2 References 13-6

13.3 PIPESYS Methods and Correlations 13-9

Index 11

1 Overview

1.1 Introduction **3**

1.2 How This Manual Is Organized **5**

1.3 Disclaimer **5**

1.4 Copyright **6**

1.5 Acknowledgements **6**

1.6 Warranty **7**

1.7 Technical Support **8**

1.7.1 Technical Support Centres 9
1.7.2 Offices 10
1.7.3 Agents 11

1.1 Introduction

A pipeline must transport fluids over diverse topography and under varied conditions. Ideally this would be done efficiently with a correctly sized pipeline that adequately accounts for pressure drop, heat losses and includes the properly specified and sized in-line facilities, such as compressors, heaters or fittings. Due to the complexity of pipeline network calculations, this often proves a difficult task. It is not uncommon that during the design phase an over-sized pipe is chosen to compensate for inaccuracies in the pressure loss calculations. With multiphase flow, this can lead to greater pressure and temperature losses, increased requirements for liquid handling and increased pipe corrosion. Accurate fluid modelling helps to avoid these and other complications and results in a more economic pipeline system. To accomplish this requires single and multiphase flow technology that is capable of accurately and efficiently simulating the pipeline flow.

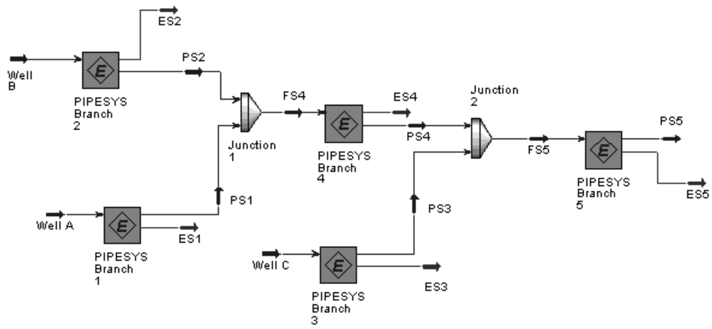
PIPESYS has far-reaching capabilities to accurately and powerfully model pipeline hydraulics. It uses the most reliable single and multiphase flow technology available to simulate pipeline flow. Functioning as an seamless extension to HYSYS, PIPESYS has access to HYSYS features such as the component database and fluid properties. PIPESYS includes many in-line equipment and facility options relevant to pipeline construction and testing. The extension models pipelines that stretch over varied elevations and environments. PIPESYS enables you to:

- rigorously model single phase and multiphase flows
- compute detailed pressure and temperature profiles for pipelines that traverse irregular terrain, both on shore and offshore
- perform forward and reverse pressure calculations
- model the effects of in-line equipment such as compressors, pumps, heaters, coolers, regulators and fittings including valves and elbows
- perform special analyses including
  - pigging slug prediction
  - erosion velocity prediction
  - severe slugging checks
- model single pipelines or networks of pipelines in isolation or as part of a HYSYS process simulation
- perform sensitivity calculations to determine the dependency of system behaviour on any parameter
- quickly and efficiently perform calculations with the internal calculation optimizer, which significantly increases calculation speed without loss of accuracy

- determine the possibility of increasing capacity in existing pipelines based on compositional effects, pipeline effects and environmental effects

Figure 1.1

A PIPESYS network:



A wide variety of correlations and mechanistic models are used in computing the PIPESYS extension. Horizontal, inclined and vertical flows may all be modelled. Flow regimes, liquid holdup and friction losses can also be determined. There is considerable flexibility in the way calculations are performed. You can:

- compute the pressure profile using an arbitrarily defined temperature profile, or compute the pressure and temperature profiles simultaneously
- given the conditions at one end, perform pressure profile calculations either with or against the direction of flow to determine either upstream or downstream conditions
- perform iterative calculations to determine the required upstream pressure and the downstream temperature for a specified downstream pressure and upstream temperature
- compute the flow rate corresponding to specified upstream and downstream conditions

Users familiar with HYSYS will recognise a similar logical worksheet and data entry format in the PIPESYS extension. Those not familiar with HYSYS will quickly acquire the skills to run HYSYS and PIPESYS using the tools available such as the user manuals, online help and status bar indicators. It is recommended that all users read this manual in order to fully understand the functioning and principles involved when constructing a PIPESYS simulation.

1.2 How This Manual Is Organized

This user manual is a comprehensive guide that details all the procedures you need to work with the PIPESYS extension. To help you learn how to use PIPESYS efficiently, this manual thoroughly describes the views and capabilities of PIPESYS as well as outlining the procedural steps needed for running the extension. The basics of building a simple PIPESYS pipeline are outlined in the Quick Start (see **Chapter 4 - Elevation Profile -Quick Start**). A more complex system is then explored in the tutorial problem (see **Chapter 10 - Gas-Condensate Tutorial**). Both cases are presented as a logical sequence of steps that outline the basic procedures needed to build a PIPESYS case. More advanced examples of PIPESYS applications are available in the Applications binder.

This manual also outlines the relevant parameters for defining the entire extension and its environment, as well as the smaller components such as the pipe units and in-line facilities. Each view is defined on a page-by-page basis to give you a complete understanding of the data requirements for the components and the capabilities of the extension.

The PIPESYS Users' Guide does not detail HYSYS procedures and assumes that you are familiar with the HYSYS environment and conventions. If you require more information on working with HYSYS, please see Volumes 1 and 2 of the ***HYSYS Reference Manual.*** Here you will find all the information you require to set up a case and work efficiently within the simulation environment.

1.3 Disclaimer

PIPESYS is the proprietary software developed jointly by Neotechnology Consultants Ltd. (hereafter known as Neotec) and Hyprotech Ltd. (hereafter known as Hyprotech).

Neither Neotec nor Hyprotech make any representations or warranties of any kind whatsoever with respect to the contents hereof and specifically disclaims without limitation any and all implied warranties of merchantability of fitness for any particular purpose. Neither Neotec nor Hyprotech will have any liability for any errors contained herein or for any losses or damages, whether direct, indirect or consequential, arising from the use of the software or resulting from the results

obtained through the use of the software or any disks, documentation or other means of utilisation supplied by Neotec or Hyprotech.

Neotec and Hyprotech reserve the right to revise this publication at any time to make changes in the content hereof without notification to any person of any such revision or changes.

1.4 Copyright

This program is protected by a hardware security device (Security Key). The authors will not be held responsible for any damage to or loss of data from the user's computer if any attempts at unauthorised copying are made.

PIPESYS™ and PIPEFLO™ are trademarks of Neotechnology Consultants Ltd.

1.5 Acknowledgements

The authors recognise all trademarks used in the manual. These include, but are not limited to, the following list:

MSDOS and Windows are registered trademarks of Microsoft Corporation. IBM is a registered trademark of International Business Machines Ltd.

Neotec and Hyprotech hereby agree to grant you a nonexclusive license to use the software program, subject to the terms and conditions set forth in the license agreement.

1.6 Warranty

Neotec, Hyprotech or their representatives will exchange any defective material or program disks within 90 days of the purchase of the product, providing that the proof of purchase is evident. All warranties on the disks and manual, and any implied warranties, are limited to 90 days from the date of purchase. Neither Neotec, Hyprotech nor their representatives make any warranty, implied or otherwise, with respect to this software and manuals.

The program is intended for use by a qualified engineer. Consequently the interpretation of the results from the program is the responsibility of the user.

Neither Neotec nor Hyprotech shall bear any liability for the loss of revenue or other incidental or consequential damages arising from the use of this product.

1.7 Technical Support

There are several ways in which you can contact Technical Support. If you cannot find the answer to your question in the manuals, we encourage you to visit our website at **www.hyprotech.com**, where a variety of information is available to you, including:

- answers to frequently asked questions
- example cases and product information
- technical papers
- news bulletins
- hyperlink to support e-mail.

You can also access Support directly via e-mail. A listing of Technical Support Centres including the Support e-mail address is at the end of this chapter. When contacting us via e-mail, please include in your message:

- Your full name, company, phone and fax numbers.
- The version of HYSYS you are using (shown in the Help, About HYSYS view).
- The serial number of your HYSYS security key.
- A detailed description of the problem (attach a simulation case if possible).

We also have toll free lines that you may use. When you call, please have the same information available.

1.7.1 Technical Support Centres

Calgary, Canada

AEA Technology - Hyprotech Ltd.
Suite 800, 707 - 8th Avenue SW
Calgary, Alberta
T2P 1H5

support@hyprotech.com (e-mail)
(403) 520-6181 (local - technical support)
1-888-757-7836 (toll free - technical support)
(403) 520-6601 (fax - technical support)
1-800-661-8696 (information and sales)

Barcelona, Spain (Rest of Europe)

AEA Technology - Hyprotech Ltd.
Hyprotech Europe S.L.
Pg. de Gràcia 56, 4th floor
E-08007 Barcelona, Spain

support@hyprotech.com (e-mail)
+34 93 724 424 (technical support)
900 161 900 (toll free - technical support - Spain only)
+34 93 154 256 (fax - technical support)
+34 93 156 884 (information and sales)

Oxford, UK (UK clients only)

AEA Technology Engineering Software
Hyprotech
404 Harwell, Didcot
Oxfordshire, OX11 0RA
United Kingdom

support@hyprotech.com (e-mail)
0800 731 7643 (freephone technical support)
+44 1235 43 4351 (fax - technical support)
+44 1235 43 4852 (information and sales)

Kuala Lumpur, Malaysia

AEA Technology - Hyprotech Ltd.
Hyprotech Ltd., Malaysia
Lot E-3-3a, Dataran Palma
Jalan Selaman ½, Jalan Ampang
68000 Ampang, Selangor
Malaysia

support@hyprotech.com (e-mail)
+60 3 4270 3880 (technical support)
+60 3 4271 3811 (fax - technical support)
+60 3 4270 3880 (information and sales)

Yokohama, Japan

AEA Technology - Hyprotech Ltd.
AEA Hyprotech KK
Plus Taria Bldg. 6F.
3-1-4, Shin-Yokohama
Kohoku-ku
Yokohama, Japan
222-0033

support@hyprotech.com (e-mail)
81 45 476 5051 (technical support)
81 45 476 5051 (information and sales)

1.7.2 Offices

Calgary, *Canada*
Tel: (403) 520-6000
Fax: (403) 520-6040/60
Toll Free: 1-800-661-8696

Yokohama, *Japan*
Tel: 81 45 476 5051
Fax: 81 45 476 3055

Newark, DE, *USA*
Tel: (302) 369-0773
Fax: (302) 369-0877
Toll Free: 1-800-688-3430

Houston, TX, *USA*
Tel: (713) 339-9600
Fax: (713) 339-9601
Toll Free: 1-800-475-0011

Oxford, *UK*
Tel: +44 1235 43 5555
Fax: +44 1235 43 4294

Barcelona, *Spain*
Tel: +34 932 724 424
Fax: +34 932 154 256

Oudenaarde, *Belgium*
Tel: +32 55 310 299
Fax: +32 55 302 030

Düsseldorf, *Germany*
Tel: +49 211 577933 0
Fax: +49 211 577933 11

Hovik, *Norway*
Tel: +47 67 10 6464
Fax: +47 67 10 6465

Cairo, *Egypt*
Tel: +20 2 720 0824
Fax: +20 2 702 0289

Kuala Lumpur, *Malaysia*
Tel: +60 3 4270 3880
Fax: +60 3 4270 3811

Seoul, *Korea*
Tel: 82 2 3453 3144 5
Fax: 82 2 3453 9772

1.7.3 Agents

International Innotech, Inc. **Katy,** *USA*	**Tel:** (281) 492-2774 **Fax:** (281) 492-8144
International Innotech, Inc. **Beijing,** *China*	**Tel:** 86 10 6499 3956 **Fax:** 86 10 6499 3957
International Innotech **Taipei,** *Taiwan*	**Tel:** 886 2 809 6704 **Fax:** 886 2 809 3095
KBTECH Ltda. **Bogota,** *Colombia*	**Tel:** 57 1 258 44 50 **Fax:** 57 1 258 44 50
KLG Systel **New Delhi,** *India*	**Tel:** 91 124 346962 **Fax:** 91 124 346355
Logichem Process **Johannesburg,** *South Africa*	**Tel:** 27 11 465 3800 **Fax:** 27 11 465 4548
Process Solutions Pty. Ltd. **Peregian,** *Australia*	**Tel:** 61 7 544 81 355 **Fax:** 61 7 544 81 644
Protech Engineering **Bratislava,** *Slovak Republic*	**Tel:** +421 7 4488 8286 **Fax:** +421 7 4488 8286
PT. Danan Wingus Sakti **Jakarta,** *Indonesia*	**Tel:** 62 21 567 4573 75/62 21 567 4508 10 **Fax:** 62 21 567 4507/62 21 568 3081
Ranchero Services (Thailand) Co. Ltd. **Bangkok,** *Thailand*	**Tel:** 66 2 381 1020 **Fax:** 66 2 381 1209
S.C. Chempetrol Service srl **Bucharest,** *Romania*	**Tel:** +401 330 0125 **Fax:** +401 311 3463
Soteica De Mexico **Mexico D.F.,** *Mexico*	**Tel:** 52 5 546 5440 **Fax:** 52 5 535 6610
Soteica Do Brasil **Sao Paulo,** *Brazil*	**Tel:** 55 11 533 2381 **Fax:** 55 11 556 10746
Soteica S.R.L. **Buenos Aires,** *Argentina*	**Tel:** 54 11 4555 5703 **Fax:** 54 11 4551 0751
Soteiven C.A. **Caracas,** *Venezuela*	**Tel:** 58 2 264 1873 **Fax:** 58 2 265 9509
ZAO Techneftechim **Moscow,** *Russia*	**Tel:** +7 095 202 4370 **Fax:** +7 095 202 4370

Internet

Website: www.hyprotech.com

E-mail: info@hyprotech.com

2 Installation

2.1 System Requirements... 3

2.2 Software Requirements 3

2.3 Installing PIPESYS 4

2.3.1 PIPESYS Extension Installation.. 4
2.3.2 Starting PIPESYS 5

2.1 System Requirements

PIPESYS has the following fundamental system requirements.

System Component	Requirement
Operating System	Microsoft Windows 2000/NT 4.0/98/95
Disk Space	Approximately 6 MB of free disk space is required.
Serial Port	The green security key is used with the standalone version of HYSYS and can only be attached to a serial communications port of the computer running the application (do not plug in a serial mouse behind the security key).
Parallel Port	SLM keys are white Sentinel SuperPro keys, manufactured by Rainbow Technologies. The Computer ID key is installed on the parallel port (printer port) of your computer. An arrow indicates which end should be plugged into the computer. This is the new key that is used for both Standalone and Network versions of HYSYS.
Monitor/Video	Minimum usable: SVGA (800x600). Recommended: SVGA (1024x768).
Mouse	Required. Note that a mouse cannot be plugged into the back of the green serial port key used with the "standalone" version of HYSYS.

2.2 Software Requirements

The PIPESYS Extension runs as a plug-in to HYSYS. That is, it is uses the HYSYS interface and property packages to build a simulation and is accessed in the same manner as a HYSYS unit operation. Therefore, to run PIPESYS you are required to have **HYSYS** - ***Version 1.2*** or higher.

Note, you will not be able to use PIPESYS without the proper HYSYS and PIPESYS licenses. You can refer to **Chapter 4 - Software Licensing** of the HYSYS Get Started Manual for information on licenses.

2.3 Installing PIPESYS

2.3.1 PIPESYS Extension Installation

The following instructions relate to installing PIPESYS as an extension to HYSYS. HYSYS must be installed prior to installing the PIPESYS Extension.

*For instructions on installing HYSYS refer to **Section 3.2 - Installing HYSYS** of the HYSYS Get Started Manual.*

1. Shut down all other operating Windows programs on the computer before starting the installation process.
2. Insert the HYSYS software CD into the CD-ROM drive of the computer.

> Note that for computers which have the CD-ROM **Autorun** feature enabled, steps #3 and #4 will be automatically performed.

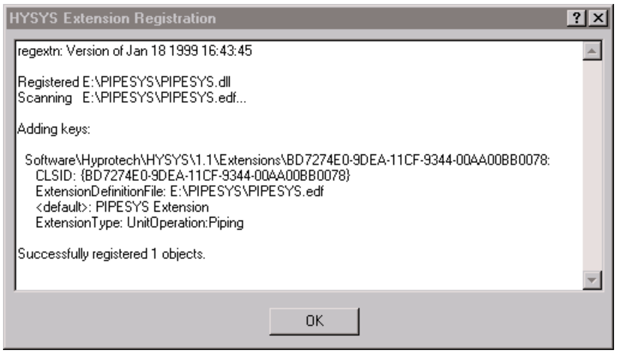
3. From the **Start Menu**, select **Run**
4. In the **Run** dialog box, type: **d:\setup.exe** and click on the **OK** button (where d: corresponds to the drive letter of the CD-ROM drive).
5. Select ‘PIPESYS’ from the following view to start the installation.

Figure 2.1

6. The first dialog that appears welcomes you to the installation program and displays the name of the application you are trying to install. If all of the information is correct click the **Next** button.
7. The following dialog provides information regarding Hyprotech’s new software security system. Please read the information presented on this screen it is important. Click the **Next** button to continue.

8. Specify a destination folder where the setup will install the PIPESYS files. If you do not wish to install the application in the default directory use the **Browse** button to specify the new path. When the information is correct click the **Next** button.
9. The installation program will then allow you to review the information that you have provided. If all of the information is correct click the **Next** button. HYSYS will then begin installing files to your computer.
10. Once the all the files have been transfered to their proper locations the installtion program will register the PIPESYS extension with HYSYS. Once the extension is successfully registered click **OK** to continue.

Figure 2.2



11. Click **FInish** to complete the installation.

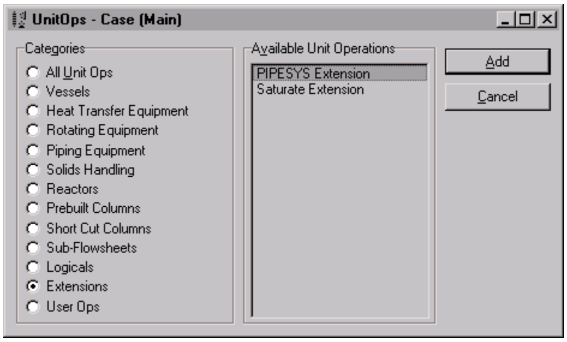
2.3.2 Starting PIPESYS

For additional information on the properties of HYSYS Unit Operations, refer to the HYSYS Steady State Modelling Manual.

You can work with PIPESYS only as it exists as part of a HYSYS case. Extensions that are part of an existing case may be accessed upon entering HYSYS' **Main Simulation Environment**. Here you can view and manipulate them as you would any HYSYS unit operation.

Before creating a new PIPESYS Extension you are required to be working within a HYSYS case that has as a minimum a Fluid Package, consisting of a property package and components. New PIPESYS Extensions are added within the Main Simulation Environment from the **UnitOps** view, which lists all the available Unit Operations.

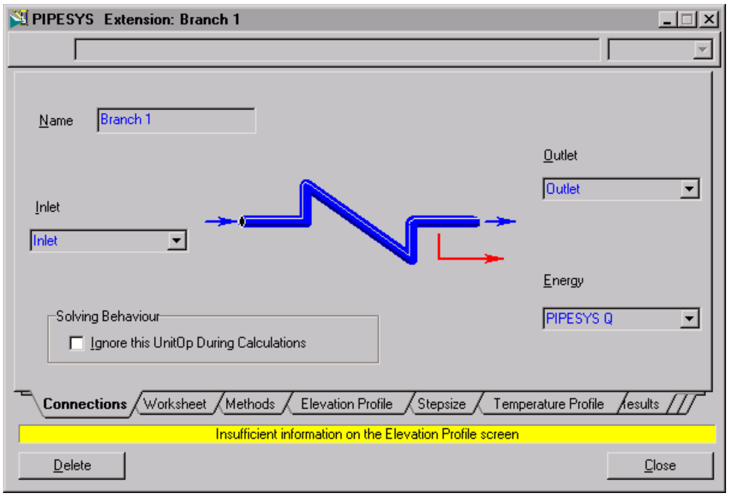
Figure 2.3



To create a new PIPESYS Extension, highlight PIPESYS Extension in the list of Available Unit Operations as shown above. Click the **Add** button and a new PIPESYS Extension will become appear on the screen.

The initial PIPESYS view is the **Connections Page** and it is shown in **Figure 2.4**.

Figure 2.4



To view any other pages of the PIPESYS view, simply click on the tab of the desired page and the view will switch to the selected page.

3 The PIPESYS View

3.1 PIPESYS Features 3

3.2 Adding PIPESYS 4

3.3 PIPESYS User Interface 8

3.4 The Main PIPESYS View 9

3.4.1 Connections Tab 10
3.4.2 Worksheet Tab 11
3.4.3 Methods Tab 11
3.4.4 Elevation Profile Tab 13
3.4.5 Stepsize Tab 18
3.4.6 Cooldown Tab 20
3.4.7 Temperature Profile Tab 23
3.4.8 Results Tab 25
3.4.9 Messages Tab 28

The **PIPESYS Extension** is a pipeline hydraulics software package used to simulate pipeline systems within the HYSYS framework. The PIPESYS Flowsheet functions in the same manner as any HYSYS unit operation or application in terms of its layout and data entry methods. The view consists of 10 worksheet tabs that may be accessed through the tabs. At the bottom of each worksheet is a status bar which guides data entry and indicates required information, as well as indicating the status of the PIPESYS simulation once the calculation has been initialized. You define the pipeline by entering pipe units and in-line facilities and specifying their length and elevation gain. By using several pipe segments, you can create a pipeline which traverses a topographically varied terrain.

PIPESYS has a comprehensive suite of methods and correlations for modelling single and multiphase flow in pipes and is capable of accurately simulating a wide range of conditions and situations. You have the option of using the default correlations for the PIPESYS calculations, or specifying your own set from the list of available methods for each parameter.

PIPESYS is fully compatible with all of the gas, liquid and gas/liquid Fluid Packages in HYSYS. You may combine PIPESYS and HYSYS objects in any configuration during the construction of a HYSYS Flowsheet. PIPESYS objects may be inserted at any point in the Flowsheet where single or multiphase pipe flow effects must be accounted for in the process simulation.

3.1 PIPESYS Features

The PIPESYS extension is functionally equivalent to a HYSYS Flowsheet Operation. It is installed in a Flowsheet and connected to Material and Energy streams. All PIPESYS extension properties are accessed and changed through a set of property views that are simple and convenient to use. Chief among these, and the starting point for the definition of a PIPESYS Operation, is the Main PIPESYS View:

- **Main PIPESYS View -** Used to define the elevation profile, add pipeline units, specify Material and Energy streams, choose calculation methods and check results.

The PIPESYS extension includes these pipeline units, each of which is accessible through a property view:

- **Pipe -** The basic pipeline component used to model a straight section of pipe and its physical characteristics.
- **Compressor -** Boosts the gas pressure in a pipeline.

- **Pump -** Boosts the liquid pressure in a pipeline.
- **Heater -** Adds heat to the flowing fluid(s).
- **Cooler -** Removes heat from the flowing fluid(s).
- **Unit X -** A “black box” component that allows you to impose arbitrary changes in pressure and temperature on the flowing fluid(s).
- **Regulator -** Reduces the flowing pressure to an arbitrary value.
- **Fittings -** Used to account for the effect of fittings such as tees, valves and elbows on the flowing system.
- **Pigging Slug Size Check -** An approximate procedure for estimating the size of pigging slugs.
- **Severe Slugging Check -** A tool for estimating whether or not severe slugging should be expected.
- **Erosion Velocity Check -** Checks fluid velocities to estimate whether or not erosion effects are likely to be significant.

3.2 Adding PIPESYS

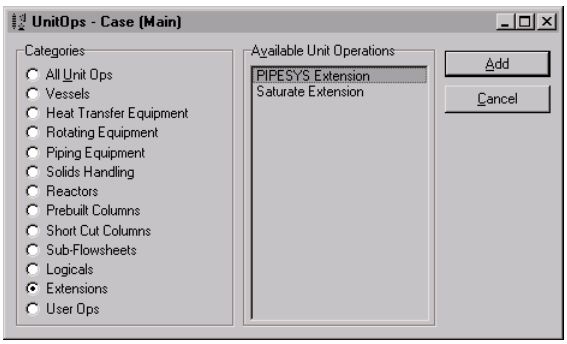
Adding a PIPESYS Extension to a HYSYS Case

Carry out the following steps to add a PIPESYS Operation to a HYSYS Case:

*For further details on creating a HYSYS case, refer to the HYSYS Reference Manual 1, **Section 1.3 - Starting a Simulation**.*

1. Your first task is to create a HYSYS Case suitable for the addition of the PIPESYS Extension. As a minimum, you must create a Case with a Fluid Package, two Material Streams and an Energy Stream.
2. With the Case open, click on the Flowsheet command from the Menu Bar and click ***Add UnitOp***. Select the **Extensions** radio button and choose the **PIPESYS Extension** from the Available Unit Operations group box on the UnitOps view. The Main PIPESYS View will open and be ready for input.

Figure 3.1

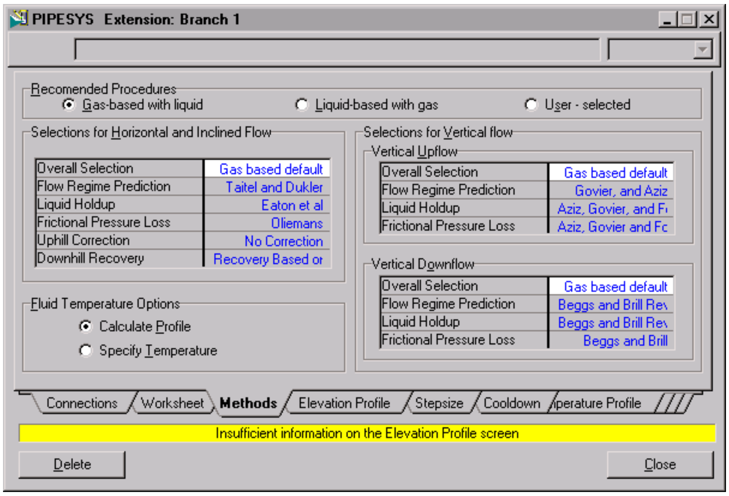


3. Select Material Streams from the Inlet and Outlet drop down lists on the **Connections** tab of the PIPESYS view. Select an Energy Stream from the Energy drop down list. If you have not yet installed these streams in the Case, they can be created by directly entering their names on the **Connections** tab. To define the stream conditions, right click on the name and select **View**.

*A gas-condensate system is a good example of a **gas-based with liquid** system because while liquid is often present, only the gas component is present under all conditions.*

4. Open the **Methods** tab. Decide on the most appropriate description of your fluid system; gas-based with liquid or liquid-based with gas. Your choice is not determined so much by the relative amounts of gas and liquid as it is by the phase that is present under all conditions of temperature and pressure in the pipeline. Select the radio button in the **Recommended Procedures** group box that corresponds to the best description of your system. If the system is determined to be single phase in the course of finding a solution, all multiphase options will be ignored.

Figure 3.2



In the **Fluid Temperature Options** group box, select either **Calculate Profile** or **Specify Temperature**. If the former is selected, the program will perform simultaneous pressure and temperature calculations, if the latter, the temperature of the fluid will be fixed according to values which you enter on the **Temperature Profile** tab and only pressure calculations will be performed.

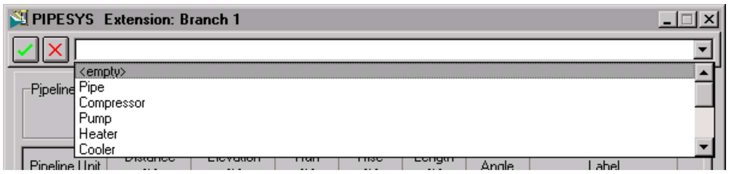
5. Define the sequence of pipeline units that make up your system on the **Elevation Profile** tab. You should start by entering values into the **Distance** and **Elevation** input cells in the **Pipeline Origin** group box; these define the position of the beginning of the pipeline, where the inlet stream is attached.

Figure 3.3

Starting with the nearest upstream unit, enter each pipeline unit by selecting the **<empty>** cell in the **Pipeline Unit** column and choosing a unit type from the drop-down list on the Edit Bar

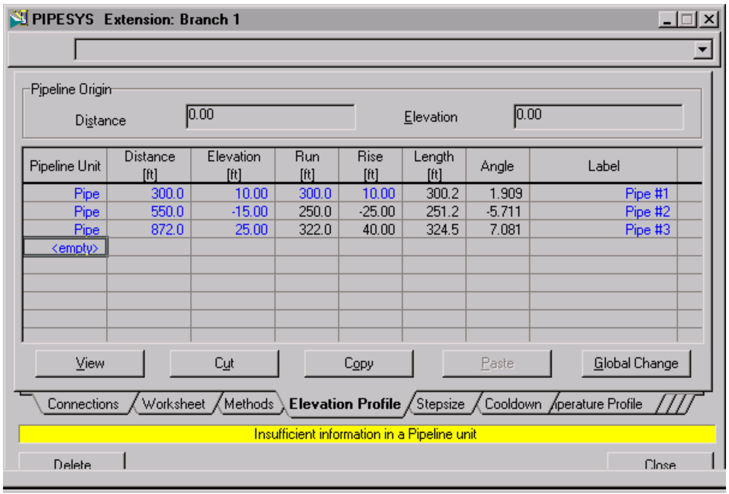
.

Figure 3.4



To insert the unit at an intermediate position rather than adding it to the end of the list, select the unit which will be immediately downstream of the new unit. Choose the unit type from the Edit Bar and the new unit will be inserted in the list, before the unit that you previously selected. A Property View for the unit will appear. You should enter all required data for the unit into this Property View before proceeding.

Figure 3.5



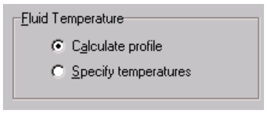
If you have added a Pipe Unit to the pipeline, you will need to define the position of the downstream end of the pipe using the **Distance**, **Elevation**, **Run**, **Rise**, **Length** and **Angle** parameters. Any two of these parameters are sufficient to fix the position of the end of the pipe. *However, if you use **Length** and one of **Run** or **Distance** to define the pipe end position, the program is unable to resolve the resulting ambiguity associated with the Angle parameter and assumes that this value should be positive.* If in fact the Angle is negative, make a note of the Angle magnitude, delete one of the Length, Distance or Run values and enter the negative of the Angle magnitude into the Angle input cell.

6. The **Stepsize** tab displays optimizing parameters used in PIPESYS algorithms. For a first-time solution of your system it is recommended that the Program Defaults radio button be selected. For most systems, the default values will provide near-optimal convergence and solution times.

7. Open the **Temperature Profile** tab. Here you can choose to specify a predetermined set of fluid temperatures for your system, as might be available from field data or if the system's sensitivity to temperature is being examined. Alternatively, you can request that the program calculate the heat transfer from the fluid to the surroundings. Select either **Calculate profile** or **Specify temperatures** in the **Fluid Temperature** group box.

*This group box is also available on the **Methods** tab.*

Figure 3.6



If you choose to specify temperatures, you must enter at least one temperature value at the Pipeline Origin. The program will use the temperature values that you do enter to fill in interpolated temperature values at each of the elevation profile points that you leave empty.

Following these steps allows you to complete the PIPESYS extension. Once the calculations are complete, as displayed by the Object Status bar, the **Results** tab will display temperature and pressure data for the pipeline and you are then able to print summary or detailed reports. The **Messages** tab reports any special problems or conditions encountered in the course of the calculations.

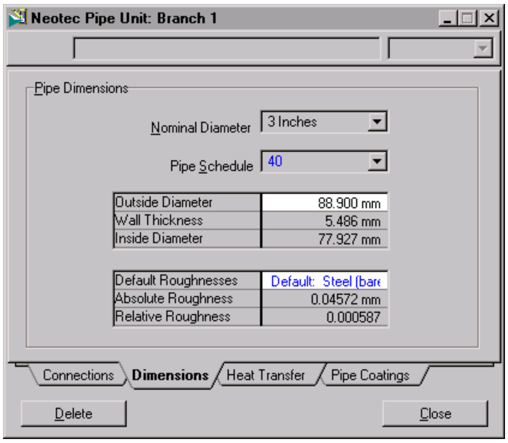
3.3 PIPESYS User Interface

The PIPESYS user interface is completely integrated into the HYSYS environment and conforms to all HYSYS usage conventions for operations and data entry. If you are an experienced user of HYSYS, you will already be familiar with all of the features of the PIPESYS user interface. If you are a new user, you should begin by studying the HYSYS Reference Manuals, since you will need to learn more about HYSYS before you can use the PIPESYS extension.

The PIPESYS user interface consists of an assortment of property views. PIPESYS Pipeline Units, of which there are many types including pipe units, pumps and compressors, are all accessible as property views. In this User's Manual, PIPESYS property views are referred to individually by the type of component they reference, so you will encounter the terms **Compressor View**, **Heater View**, **Fittings View** etc.

Like all HYSYS property views, PIPESYS property views allow access to all of the information associated with a particular item. Each view has a number of tabs and on each tab are groups of related parameters. For example, on the Dimensions tab of the Pipe Unit View (See Figure 3.7) the physical characteristics of the Pipe Unit, such as wall thickness, material type and roughness can be specified.

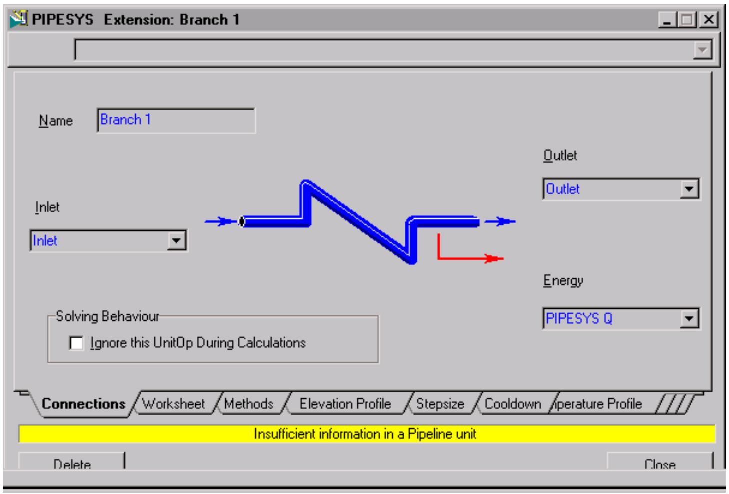
Figure 3.7



3.4 The Main PIPESYS View

The Main PIPESYS View is the first view that appears when adding a PIPESYS operation to a HYSYS Flowsheet. This view provides you with a place to enter the data that defines the basic characteristics of a PIPESYS operation. Here you can specify pipeline units, elevation profile data, calculation procedures, tolerances and all other parameters common to the PIPESYS operation as a whole.

Figure 3.8



The Main PIPESYS View is the starting point for the definition of any PIPESYS operation. When you select **Flowsheet/Add Operation**... from the Menu Bar and then choose PIPESYS extension, the Main PIPESYS View will appear and be ready to accept input. You must then select each of the tabs on the Main PIPESYS View and complete them as required.

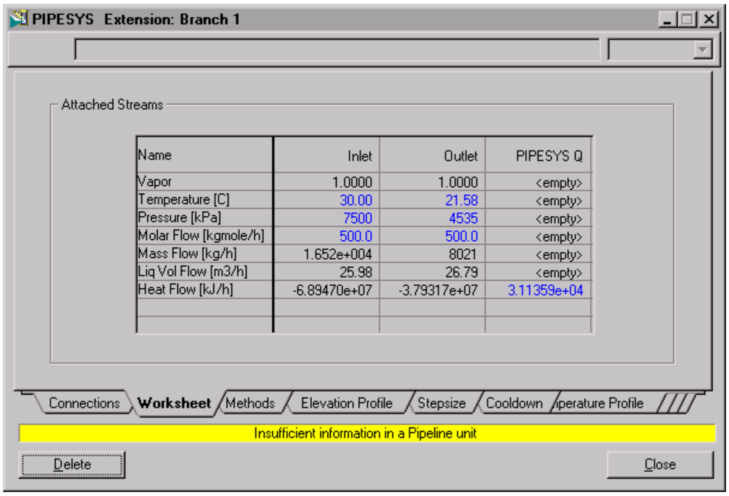
3.4.1 Connections Tab

This tab is used to define the connections between the HYSYS simulation case and the PIPESYS operation. The inlet, outlet and energy streams are specified here using the **Inlet**, **Outlet** and **Energy** drop down input cells. You may also choose a name for the operation and enter this in the **Name** input cell. The **Ignore this UnitOp During Calculations** check box can be selected if you wish to disable the concurrent calculation of intermediate results during data entry. This setting is recommended if you have a slow computer and data processing is slowing down the entry process or if you wish to delay the calculations until you have entered all of your data.

3.4.2 Worksheet Tab

This tab allows you to directly edit the **Material** and **Energy Streams** that are attached to the PIPESYS operation without having to open their **Property Views**.

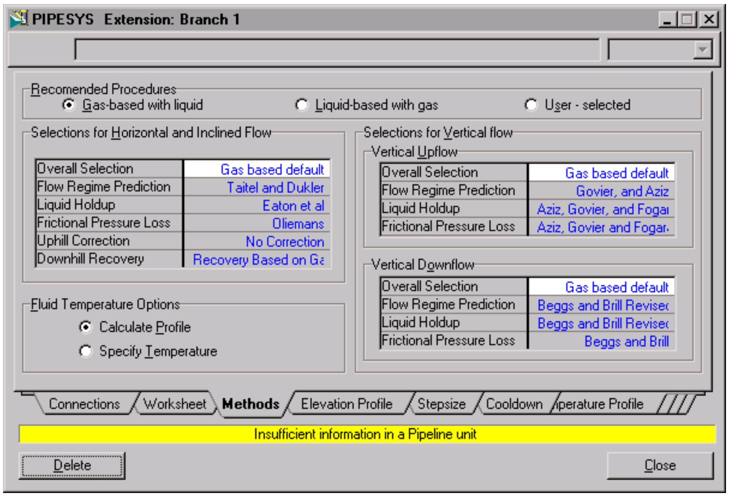
Figure 3.9



3.4.3 Methods Tab

Many correlations and models have been developed by researchers to perform multiphase flow calculations. PIPESYS makes many of them available to you on the **Methods** tab. Completion of this tab can be a simple matter of selecting one of the two fluid system classifications and allowing PIPESYS to automatically choose the calculation methods. Alternatively, if you are familiar with multiphase flow technology, you are able to specify which methods should be used.

Figure 3.10



*Examples of **gas-based systems** include dry gas, gas-condensate and gas water systems.*

*Examples of **liquid-based systems** include hydrocarbon liquid, crude oil and oil-gas systems.*

Effective use of the settings on this tab requires you to correctly classify the fluid system as being either **gas-based with liquid** or **liquid-based with gas**. A gas-based system has a gas phase that is present under all conditions and there may or may not be a liquid phase. Conversely, a liquid-based system has a predominant liquid component. The liquid component will be present under all conditions and the gas phase may or may not be present. If the software detects that only a single-phase is present in the stream (i.e. pure water, dry gas), all multiphase options are ignored and pressure loss is computed using the Fanning equation.

If the vertical or horizontal orientation of a pipeline unit is such that you have a preference for a particular calculation method, you are able to select it on this tab. For instance, if the prediction of liquid hold-up in a pipeline is a particular concern, you can manually select **OLGAS** to perform this calculation instead of using the default method. However, it is not advised to change the default settings unless you have reason to believe that a different calculation method will yield more accurate results. Generally, the safest procedure will be to use radio buttons in the Recommended Procedures group box to select either Gas-based with liquid or Liquid-based with gas, whichever classification best describes the system under consideration. PIPESYS will then set all of the selections for the various types of flows to those methods that will give the most consistent results.

In the **Fluid Temperature Options** group box, select either **Calculate Profile** or **Specify Temperature**. If the former is selected, the program will perform simultaneous pressure and temperature calculations. If the latter, the temperature of the fluid will be fixed according to values which you enter on the **Temperature Profile** tab and only pressure calculations will be performed.

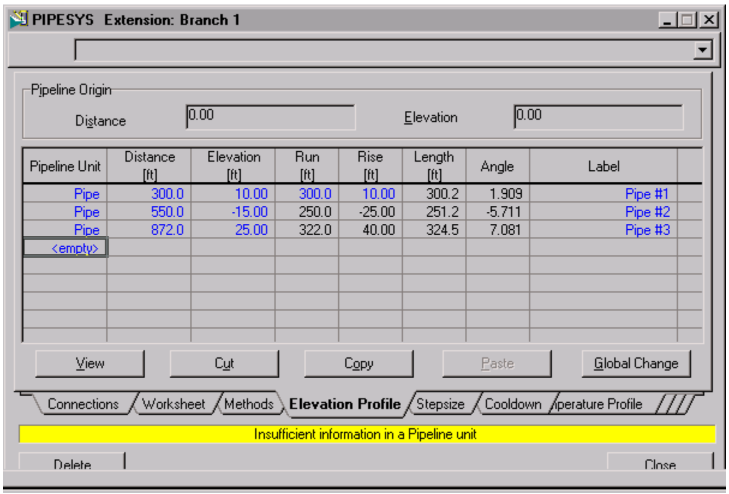
PIPESYS attempts to protect against improper usage of calculation methods. Certain combinations of methods are disallowed if there are incompatibilities and PIPESYS will display a warning message if such a combination is selected. However, there are many situations where a number of methods are valid but where some of these will give more accurate results than others for a given case. Some methods tend to give consistently better results than others for particular fluid systems. PIPESYS has been designed to default to such methods for these cases.

3.4.4 Elevation Profile Tab

The Pipeline Origin defines the point at which the inlet stream connects with the PIPESYS extension.

On this tab, the components and geometry of the pipeline system are defined. A starting point for the profile must be specified at the top of the tab in the **Pipeline Origin** group box, using the **Distance** and **Elevation** input cells. The starting point for the profile can have negative, zero, or positive distance and elevation values, but the position represented by these values must correspond to the point connected to the inlet stream of the PIPESYS extension.

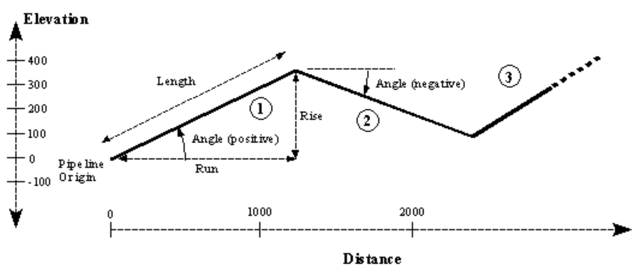
Figure 3.11



When defining the geometry of the pipeline, you must be aware of the distinction between the two types of components. The set of pipeline components in PIPESYS collectively known as **Pipeline Units** includes both **Pipe Units**, which are straight sections of pipe, and **In-line Facilities**, which are pieces of equipment such as **compressors**, **pumps**, **fittings** and **regulators**. **Pipe Units** have a starting point and an ending point and occupy the intervening space but in-line facilities are considered to occupy only a single point in the pipeline.

When a Pipe Unit is added to the pipeline, the data required to fix the position of its starting point and its ending point must be specified. The starting point of the Pipe Unit is generally already determined, since the Pipe Unit is attached to the previous unit in the pipeline. All that remains is to enter the data that PIPESYS needs to fix the end point, which can be done in a number of ways. You can fill in the **Distance** and **Elevation** cells, which define the end point of the Pipe Unit relative to the Pipeline Origin. Alternatively, you can use some combination of the **Run**, **Rise**, **Length** and **Angle** values to fix the end point relative to the Pipe Unit's starting point. For instance, you could enter a value of -10^o in the Angle cell and 300 ft in the Run cell to fix the end point as being at a horizontal distance of 300 ft from the starting point and lying on a downward slope of 10^o.

Figure 3.12



The first three segments of a pipeline elevation profile and the parameters that are used to define its geometry.

If you enter values into Length and one of Distance or Run, the PIPESYS assumes that Angle is positive. If Angle is actually negative, record the calculated Angle or Rise value, delete the contents of the Length cell and enter the negative of the recorded Angle or Rise value into the respective cell.

The Elevation Profile parameters used to define Pipe Unit endpoints are defined as follows:

- **Distance -** The horizontal position of the endpoint of the Pipe Unit, using the Pipeline Origin as the reference point.
- **Elevation -** The vertical position of the endpoint of the Pipe Unit, using the Pipeline Origin as the reference point.
- **Run -** The horizontal component of the displacement between the starting point and the ending point of a Pipe Unit.
- **Rise -** The vertical component of the displacement between the starting point and the ending point of a Pipe Unit.
- **Length -** The actual length of the Pipe Unit, measured directly between the starting point and the ending point.
- **Angle -** The angle formed between the Pipe Unit and the horizontal plane. This value will be negative for downward sloping Pipe Units and positive for upward sloping Pipe Units.

Adding an in-line facility to the pipeline is simpler because a single point is sufficient to fix its location. In most cases, you will not have to supply any location data because the position of the in-line facility will be determined by the endpoint of the previous Pipeline Unit.

Entering the Elevation Profile

Enter the Pipeline Units into the Elevation Profile in the order that they appear in the flow stream.

The elevation profile matrix on this tab provides a place for you to enter the sequence of Pipeline Units and the data that defines the geometry of the profile. You must enter the Pipeline Units in the order in which they appear in the flow stream, so that the first entry is the unit connected to the inlet stream and the last entry is the unit connected to the outlet stream. A Pipeline Unit can be entered as follows;

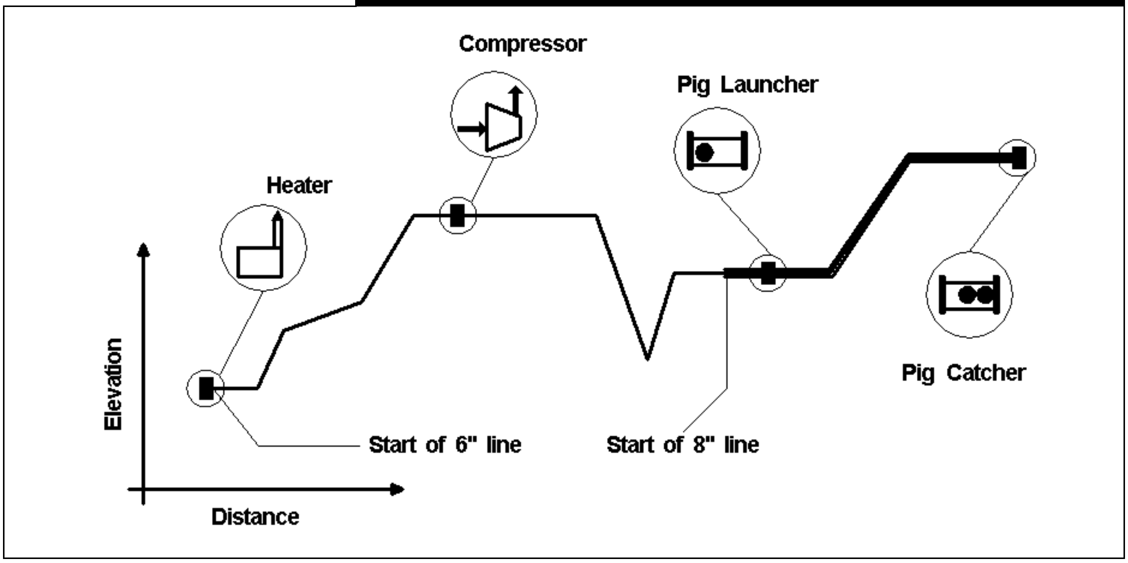
1. Select the cell with <**empty**> in it to place the new unit at the end of the sequence. To place the new unit at some other point in the sequence, select the unit that you want the new unit to precede.
2. From the drop down list on the **Edit Bar**, select the **Pipeline Unit** of the type that you want to add to the sequence. A new unit will be immediately added to or inserted in the matrix.
3. Now complete the location data if you have entered a **Pipe Unit**. You will have to define *at most two* of the **Distance**, **Elevation**, **Run**, **Rise**, **Length** or **Angle** quantities. The remaining cells will be filled in automatically once PIPESYS has enough information to complete the specification. For instance, entering the **Distance** and **Elevation** data will result in the **Run**, **Rise**, **Length** and **Angle** cells being filled in since all of these quantities can be calculated from a knowledge of the start and end points of the Pipe Unit. If

you are entering an in-line facility, the location will be filled in automatically as the program will obtain this data from the previous Pipeline Unit.

4. Optionally, provide PIPESYS with a **Label** entry. This is used by the PIPESYS program to uniquely identify each Pipeline Unit during calculations and for displaying error messages for a particular unit. The program will automatically generate a default label but you may change this if you wish. There is no restriction on the number of characters used for this label except that you may wish to use only as many as are visible at once in the cell.

The entire pipeline from the inlet to the outlet is thus described as a connected sequence of Pipeline Units. Some of these units can be pipe segments of constant slope, called Pipe Units, while others can be in-line facilities, such as compressors, pumps, heaters and fittings.

Figure 3.13



To make data entry easier for successive units, especially when most of the properties remain unchanged from unit to unit, make use of the ***Cut*** and ***Paste*** or the ***Copy*** and ***Paste*** functions. These buttons will copy the contents of the current Pipeline Unit to memory so that all the data they contain (i.e. pipe diameter for the Pipe Unit) can then be copied to a new Pipeline Unit. The **Cut** operation will copy data to memory before removing the unit, whereas the copy function will make a copy and preserve the original unit. The **Paste** operation will create a new Pipeline Unit at the cursor position. As explained above, if this is a Pipe Unit, it will then be necessary to enter any two of distance, elevation, run, rise, length or angle.

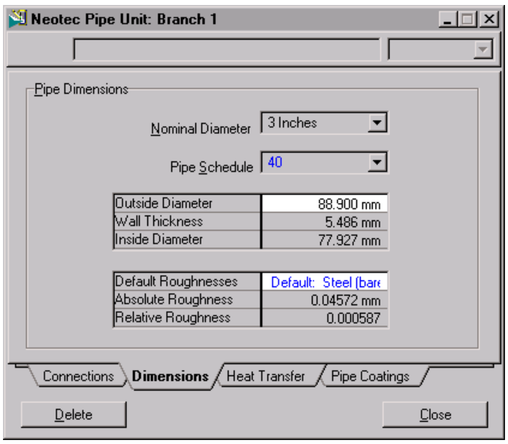
The ***Global Change*** button allows you to change the parameters for several or all of the **Pipe Units** in the Elevation Profile. This feature has been implemented in PIPESYS as a time saving mechanism so that if the same information is required for several Pipe Units, you do not need to open the Property Views for each individual Pipe Unit and to change the data. A global change operation simultaneously accesses any or all of the Pipe Units in the elevation profile and can change a selection of parameters.

For example, having made a pressure drop calculation for a 4" pipeline, you may want to repeat the calculation for the same pipeline using 6" pipe. Using the Global Change feature, you could in a single procedure change the pipe diameters from 4" to 6" for all **Pipe Units**.

The **Global Change** feature can be used to edit the **Property View** parameters for a single Pipe Unit and to subsequently duplicate the edits for none, some or all of the other Pipe Units in the pipeline, in a single sequence of operations. Any Pipe Unit can be used as a data template for changing the other Pipe Units in the pipeline.

To implement a global parameter change for some or all of the Pipe Units in the elevation profile, select any one of the Pipe Units in the elevation profile matrix and press the ***Global Change*** button. The Global Change Property View will appear. This Property View is identical to the Pipe Unit Property View except that it has check boxes beside each of the major data types on each of its tabs.

Figure 3.14



These check boxes have two functions:

1. they become checked automatically when you change a parameter to remind you that a particular parameter has been selected for a global change, and
2. you can check them manually to indicate to the program that a particular parameter will be copied to other Pipe Units using the Global Change feature.

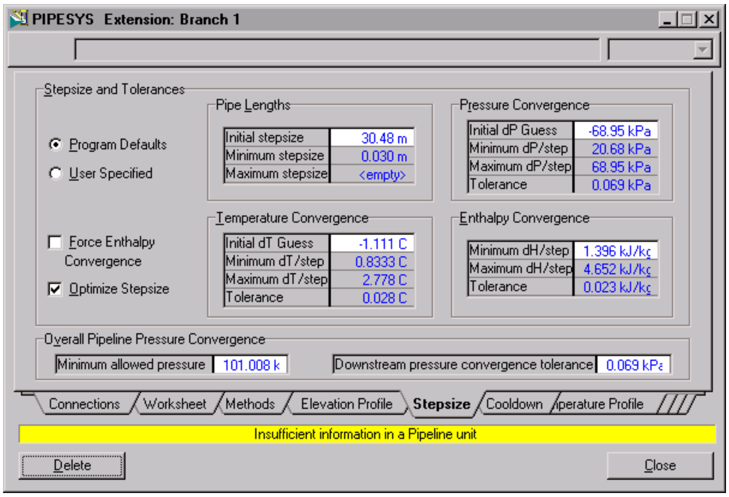
*For more information on making global changes, see **Chapter 6 - Global Change Feature**.*

Request a Global Change for a particular parameter by entering the new parameter values into the input cells. Once you have entered all the changes that you want to make, press the ***Apply*** button and the Global Change dialog box will appear with a list of all the Pipe Units in the profile. Select the Pipe Units in the list that will be included in the global change and press the ***OK*** button. The program will then make the specified parameter the changes to all of the Pipe Unit parameters that were checked.

3.4.5 Stepsize Tab

PIPESYS computes the change in pressure due to friction, hydrostatic head and kinetic energy and the change in temperature for the flowing fluid(s). These calculations are dependent on the physical characteristics and orientation of the pipe and its surroundings. They are also dependent on the fluid properties (i.e. density, viscosity, enthalpy, phase behaviour, etc.). Since these properties change with pressure and temperature, it is necessary to choose some interval over which the average properties can be applied to the calculations (i.e. a calculation length, or step, sufficiently small for property changes to be nearly linear).

Figure 3.15



To safeguard against a step size that is too large, PIPESYS has input cells containing the Maximum dP per step or Maximum dT per step. If this pressure change (dP) or temperature change (dT) is exceeded on any calculation, the step size is halved and the calculations repeated. An arbitrarily small step size could perhaps be chosen by the software to meet these criteria, but this could result in greatly increased run time with no corresponding increase in accuracy. Defaults are provided for these parameters and you will rarely be required to change them.

There may be cases where you wish to enter your own stepsize values. For this reason, you will find cells on this tab where you can not only specify an initial step size, but where you can also enter maximum and minimum allowed pressure and temperature changes. Checking the ***Stepsize Optimizer*** check box then requests that PIPESYS determine the stepsize such that the pressure/temperature changes fall within the specified maximum and minimum. As well, a minimum and maximum stepsize can be entered to constrain the optimizer.

Since the relationship between fluid properties and pressure/temperature change is implicit, PIPESYS performs an iterative calculation of pressure and temperature change at each of the steps mentioned above. Initial guesses for the change in pressure, temperature or enthalpy can be specified or left as program defaults. For multiple component multiphase systems, iterations converge on

pressure and temperature. For single component multiphase systems, or systems which behave in a similar way, iterations converge on pressure and enthalpy. Pressure, temperature and enthalpy convergence can be controlled by your input for convergence tolerance. If PIPESYS encounters difficulty in converging to a solution, perhaps due to unusual fluid property behaviour, you should try to repeat the calculation with the ***Force Enthalpy Convergence*** check box selected. This approach requires more computer time, but may succeed where the temperature convergence fails.

The **Minimum Allowed Pressure** in the cell at the bottom left controls the point at which PIPESYS will terminate the calculations due to insufficient pressure. *The program default is one atmosphere.*

When the case is such that PIPESYS is required to compute pressure at the inlet of the pipeline given a fixed downstream pressure, an iterative procedure is performed over the entire pipeline. Calculations proceed until the calculated downstream pressure converges to the fixed downstream pressure within some tolerance, specifically the **Downstream Pressure Convergence Tolerance**.

3.4.6 Cooldown Tab

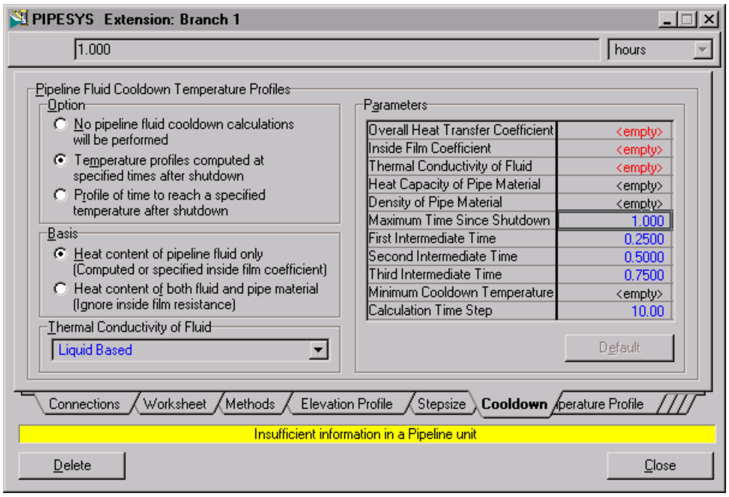
In pipelines that are used to transport a relatively high pour point crude oil, or a gas system that is subject to hydrate formation, it is usually necessary to maintain a minimum flowing temperature to avoid excessive pressure losses or even line blockage. Such pipelines are often insulated and may have one or more heaters.

When one of these pipelines is shutdown for an extended period of time, it must generally be flushed or vented to remove the hydrocarbon fluid, since the temperature in the system will eventually come to equilibrium with the surroundings. Apart from the time and effort involved in this operation, the subsequent re-starting of the pipeline is more complicated after it has been purged than if it could be simply be left filled with the original hydrocarbon fluid.

In the case of an emergency shut down, however, it may be possible to carry out whatever remedial action is required before the temperature reaches the minimum allowable value. In such cases, the line can be re-started much easier than if it has been purged and it is thus of interest to be able to predict, with reasonable accuracy, how long the fluid will take to cool down to any particular temperature.

This is, of course, a complex transient heat transfer problem (especially for multiphase fluid systems) and a rigorous solution is generally not possible. The cooldown calculations in PIPESYS should however provide approximate answers that should be capable of reasonable accuracy in many cases of interest.

Figure 3.16



The option to do cooldown calculations can be enabled on the Cooldown tab of the PIPESYS Extension's Main View when the flowing fluid temperature profile is calculated. There are two fluid temperature cooldown options that you may choose from.

- Temperature profiles computed at specified times after shutdown
- Profile of time to reach a specified temperature after shutdown

For both of the above options, the calculations can be based on one of two options.

- Heat content of the pipeline fluid only (Computed or specified inside film heat transfer coefficient)
- Heat content of both the fluid and pipe material (Ignoring the inside film heat transfer coefficient).

For calculations are based on the heat content of the pipeline fluid only (computed or specified inside film heat transfer coefficient) the fluid thermal conductivity, inside film coefficient or overall heat transfer coefficient can either be specified or computed by the program. If the

overall heat transfer coefficient is specified the option to specify the inside film heat transfer coefficient no longer exists.

For calculations based on the heat content of both the fluid and pipe material (ignoring the inside film heat transfer coefficient) the overall heat transfer coefficient can either be specified or computed by the program. Both the heat capacity of the pipe material and the density of the pipe material must be specified and defaults are available for these parameters.

Both of the calculations, based on either the heat content of the pipeline fluid only or the heat content of both the fluid and pipe material, allow the fluid thermal conductivity to be specified or calculated at all times (unless the overall heat transfer coefficient is specified). The fluid thermal conductivity can be calculated based on the liquid, gas or blended thermal conductivities. By default the calculations use the liquid thermal conductivity as this presents the most conservative results for both calculated times and temperatures. As a note, the fluid thermal conductivity is not used by the calculations when the inside film heat transfer coefficient is specified unless a Pipe Unit has its overall heat transfer coefficient specified.

The option to compute temperature profiles at specified times after shutdown requires that the:

- maximum
- first
- second
- and third intermediate times since shutdown be entered.

The intermediate times must be in increasing order and less than the maximum time. Defaults are available for these times whereby the first, second and third intermediate times are set to be one quarter, one half and three quarters of the maximum time since shutdown respectively.

The profile of time required to reach a specified temperature after shutdown requires that the minimum cooldown temperature be entered.

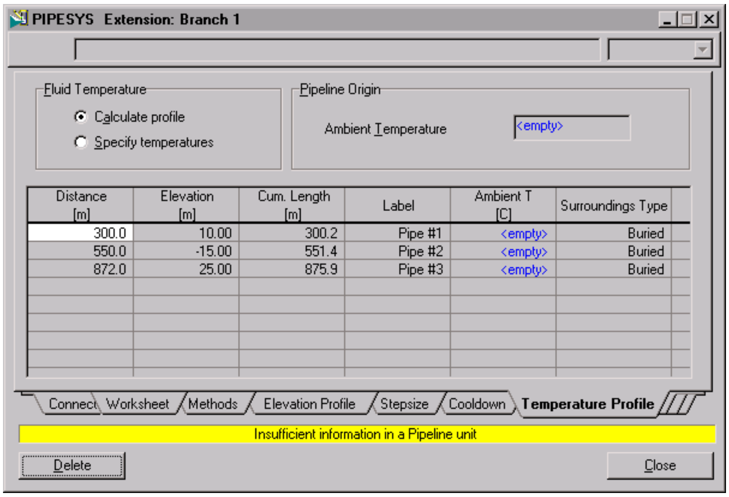
Both of the options available for the cooldown calculations require the calculation time step to be entered. A default value of ten minutes is provided as a reasonable value for this parameter.

3.4.7 Temperature Profile Tab

This tab allows you to select one of two options for handling fluid temperature effects in the pipeline. The **Fluid Temperature** group box in the top left corner is also located on the **Methods** tab and it is included here only as a matter of convenience, should you wish to change your initial selection.

To compute the pipeline pressure profile, PIPESYS must know the fluid property behaviour, and must therefore know the temperature of the fluids at every calculation point in the pipeline. You can enter the temperature directly if known, or if you are testing the sensitivity of the pipeline to temperature effects.

Figure 3.17

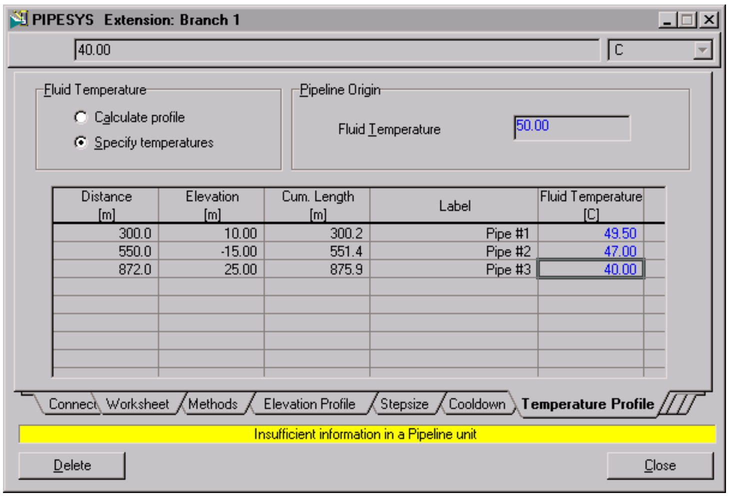


Alternatively, you can request detailed heat transfer calculations. Pipe surroundings and heat transfer parameters are entered in each Pipe Unit View while creating the pipeline elevation profile. The surroundings type for each Pipe Unit is displayed here as an overview of the system for verification purposes. If you choose to switch from a specified temperature profile to a calculated profile, note that the **Pipe Units** will have to be updated with data for heat transfer calculations not previously required. In this case, PIPESYS will warn you of missing data when calculations are attempted. Reasonable default values will be made available for unknown data.

To enter the pipeline fluid temperatures directly, select the ***Specify Temperatures*** *radio button in the* ***Fluid Temperature*** *group box.*

To enter temperatures directly, select the ***Specify Temperatures*** radio button. The matrix will display the profile previously entered on the Elevation Profile tab. In the **Fluid Temperature** column, you can enter the flowing temperature at the end of each Pipe Unit. You must enter at least one flowing temperature at the start of the pipeline and this value is entered in the **Fluid Temperature** input cell in the **Pipeline Origin** group box. All other temperatures are entered in the Fluid Temperature column of the appropriate Pipe Unit. For any cells that are empty between specified temperatures, PIPESYS will interpolate linearly the flowing temperatures (enter only one more fluid temperature in the last cell of the profile to automatically create a linear profile). For any cells that are empty after the last entered temperature, PIPESYS will assume the flowing temperature to be isothermal and will fill in the cells with a constant temperature equal to the last entered temperature. You can easily overwrite a cell with your own value anywhere the software has filled in a temperature for you.

Figure 3.18

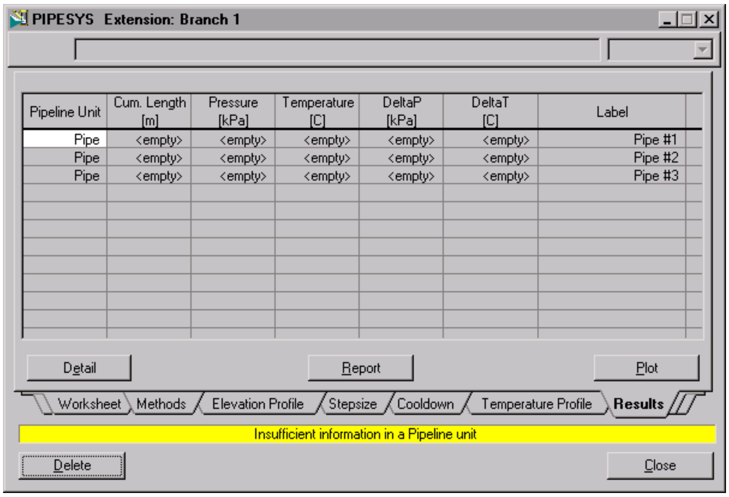


PIPESYS calculates the fluid temperature when the ***Calculate Profile*** button in the **Fluid Temperature** group box is selected. Much like the specified temperatures, you must enter at least one temperature value of the surroundings into the **Ambient Temperature** input cell in the **Pipeline Origin** group box. Any other values can be entered in the **Ambient T** column corresponding to the surroundings temperature at the end of a pipe segment. For any empty cells between the origin and a Pipe unit with a surroundings temperature, PIPESYS will interpolate linearly and fill them in with calculated values. Any other cells that are empty will be filled with the last entered temperature. As with the specified temperatures, you can overwrite any of the filled-in ambient temperature cells.

3.4.8 Results Tab

Calculation results at the endpoint of each pipeline unit are summarized on the screen in columns of **Pressure**, **Temperature**, **Pressure Change** and **Temperature Change**. The length and label as entered in the **Elevation Profile** tab are also displayed.

Figure 3.19



The **Results** tab also features the ***Detail*** button, the ***Report*** button, and the ***Plot*** button. These buttons give you the capability to view your data and results in a number of formats.

If you want to see results in greater detail than are displayed on the Results tab matrix, press the ***Detail*** button. This will bring up the **Pipe Segment Results** dialogue box that displays detailed results for each calculation step. The pipe segment for each step is controlled according to the parameters on the **Step Size** tab. For each of these the **Pipe Segment Results** dialogue box displays:

- **Sum. Length -** Horizontal distance of the Segment from the Pipeline Origin.
- **Inside Diameter -** Inside diameter of the pipe over the length of the Segment.
- **Pressure -** The fluid pressure at the downstream end of the Segment.
- **Temperature -** The fluid temperature at the downstream end of the Segment.

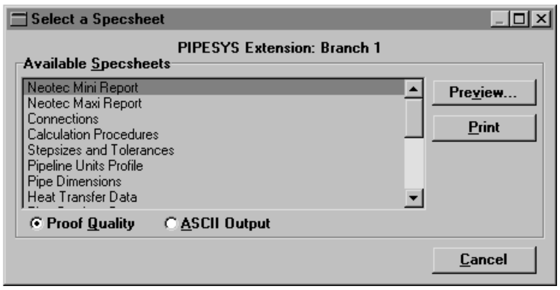
- **DeltaP Friction -** The pressure loss across the Segment due to friction.
- **DeltaP Head -** The loss or gain in the elevation head across the Segment.
- **Liq. Volume Fraction -** The volume fraction of the fluid in the Segment in the liquid phase.
- **Press. Gradient -** The pressure change per unit of pipe length.
- **Iterations -** The number of times that the program repeated the solution algorithm before convergence was obtained.
- **Gas Density -** The average density of the gas phase in the Segment.
- **Liquid Density -** The average density of the liquid phase in the Segment.
- **Gas Viscosity -** The average viscosity of the gas phase in the Segment.
- **Liquid Viscosity -** The average viscosity of the liquid phase in the Segment.
- **Vsg -** The average superficial velocity of the gas in the Segment
- **Vsl -** The average superficial velocity of the liquid in the Segment
- **Flow Pattern -** When multiphase flow occurs, the flow pattern or flow regime in a Segment is classified as being one of the following types: Stratified, Wave, Elongated Bubble, Slug, Annular-Mist, Dispersed Bubble, Bubble, or Froth. When the fluid system is in single phase flow, Single Phase is reported here.
- **Surface Tension -** The liquid property caused by the tensile forces that exist between the liquid molecules at the surface of a liquid/gas interface**.**

*For more information on using the HYSYS Report Manager and Report Builder see Section 6.2 - **Reports** in the HYSYS Reference Manual.*

PIPESYS Specsheets are available to the **HYSYS Report Manager** and can be added to a Report using the **Report Builder.**

You can also preview and print PIPESYS **Specsheets** directly from the **Results** tab. Press the ***Report*** button to bring up the Select a Specsheet dialogue. Here you can choose from a number of different Specsheets:

Figure 3.20

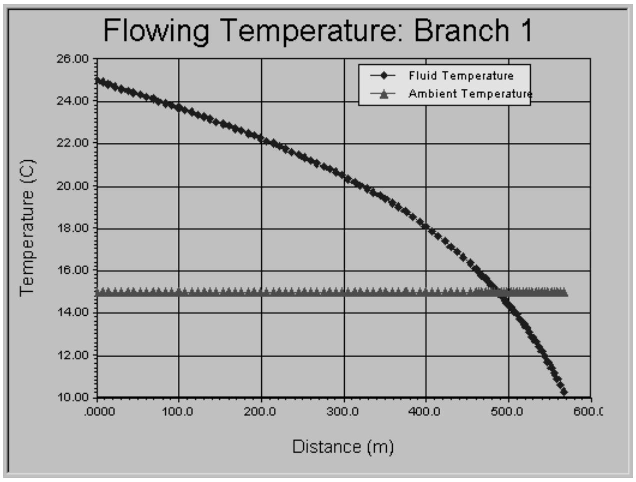


The **Neotec Mini Report** provides a summary of selections and results from the PIPESYS case. The **Neotec Maxi Report** includes the same information as the Mini Report and has additional detailed calculation results for the Pressure and Temperature profiles and Fluid Transport properties. Press the ***Preview*** button to view the formatted Specsheet on the screen, or press the ***Print*** button to print it directly.

If you do not need the complete report results from the PIPESYS case and are interested in only one particular aspect of the case, select a Specsheet that confines itself to reporting the parameter of interest. For example, select the Pressure Temperature Summary for a record of the pressure and temperature at each of the Pipeline Units.

The ***Plot*** button allows you to view your data and results in graphical form, such as the one in Figure 3.20. Press the ***Plot*** button to display the Plot view. Display any of the plots listed on the left-hand side by selecting the corresponding radio button. The initial size of the plot may be to be too small, so press the ***Pin*** button to convert the view to a Non-Modal state and press the Maximize button. To print the plot, right-click anywhere in the plot area and a pop-up menu will appear; you can then select ***Print Plot***.

Figure 3.21



Where two quantities are traced, a plot legend is displayed on a yellow rectangular background. If this obscures a plot line it can be moved by double-clicking in the plot area. This action selects the plot area to be modified and you can then drag the plot key to another location.

For more information on the Graph Control, see HYSYS Reference Manual 1, ***Section 5.3 - Graph Control****.*

To modify the characteristics of the plot, right-click on the plot area and select **Graph Control** from the pop-up menu that appears. The Graph Control tool allows you to change the Data, Axes, Title, Legend and Plot Area. For example, you can change the scaling on the plot axes by opening the Axes tab, selecting the variable to be re-scaled in the list of axes and removing the check from the Use Auto-Scale check boxes in the Bounds group box. Then change the values in Minimum and Maximum input boxes. When the ***Close*** button is pressed, the plot will be redrawn with the new scales.

3.4.9 Messages Tab

The text window of this tab is used to display messages or warnings that may have arisen during the PIPESYS extension calculations.

4 Elevation Profile - Quick Start

4.1 Flow Sheet Set-Up 3

4.2 Adding the PIPESYS Extension 4

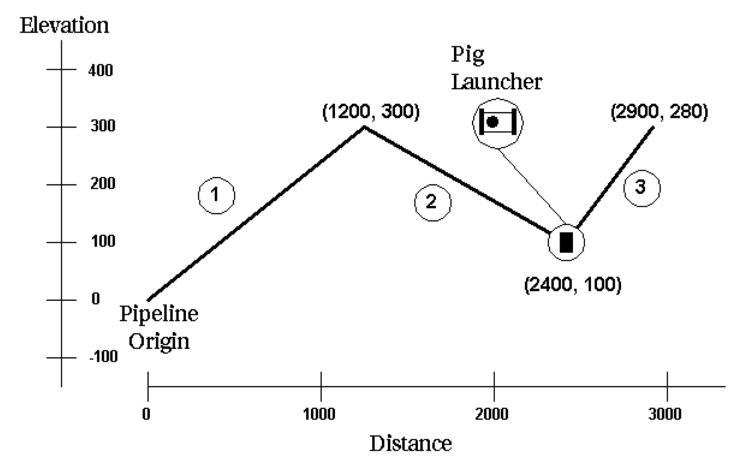
4.3 Defining the Elevation Profile 5

If you would like to follow a more detailed step-by-step procedure for creating a PIPESYS case, see ***Chapter 10 - Gas-Condensate Pipeline.***

One of the first and most important steps in adding a PIPESYS operation to a HYSYS Flowsheet is the construction of the elevation profile. The purpose of this procedure is to create a representation of the pipeline as a connected series of components with the corresponding position data. In this example, you will go through the steps to enter an elevation profile components and data. All units of measurement in this example are SI, but feel free to change these to whatever unit system you are accustomed to using

For this case, a simple pipeline consisting of three pipe units and a pig launcher will be built to demonstrate the PIPESYS procedures. Figure 4.1 shows a schematic of these four components with coordinate axes.

Figure 4.1



4.1 Flow Sheet Set-Up

Before working with the PIPESYS extension, you must first create a HYSYS case. In the Simulation Basis Manager, create a fluid package using the **Peng Robinson** equation of state. Add the components **methane, ethane, propane, i-butane, n-butane, i-pentane, n-pentane, n-hexane, nitrogen, carbon dioxide and hydrogen sulfide**.

Property Package	Components
Peng Robinson	C1, C2, C3, i-C4, n-C4, i-C5, n-C5, C6, Nitrogen, CO2, H2S

Create a stream called **Inlet** in the Main Simulation Environment and define it as follows:

*** signifies required input*

Name	Inlet
Vapour Fraction	1.00
Temperature [^{o}C]	45**
Pressure [kPa]	8000**
Molar Flow [kgmole/h]	300**
Mass Flow [kg/h]	6595
LiqVol Flow [m3/h]	17.88
Heat Flow [kJ/h]	-2.783e+07
Comp Mass Frac [methane]	0.7822**
Comp Mass Frac [ethane]	0.0803**
Comp Mass Frac [propane]	0.0290**
Comp Mass Frac [i-Butane]	0.0077**
Comp Mass Frac [n-Butane]	0.0246**
Comp Mass Frac [i-Pentane]	0.0074**
Comp Mass Frac [n-Pentane]	0.0072**
Comp Mass Frac [n-Hexane]	0.0012**
Comp Mass Frac [Nitrogen]	0.0098**
Comp Mass Frac [CO2]	0.0409**
Comp Mass Frac [H2S]	0.0097**

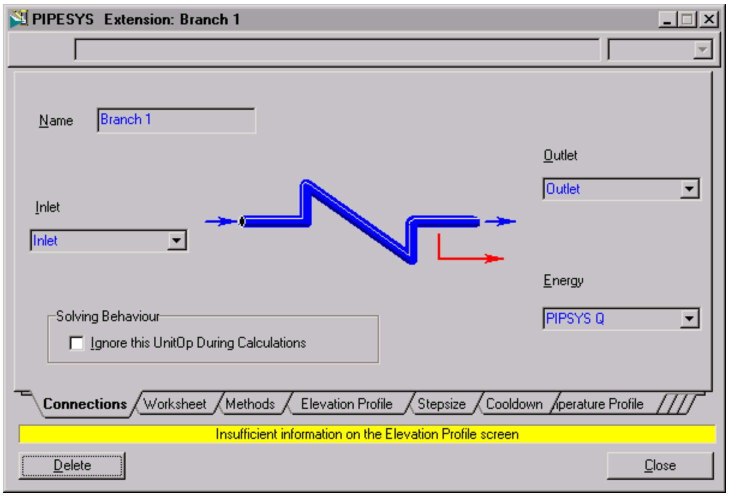
4.2 Adding the PIPESYS Extension

Once the case is created, the PIPESYS extension can be added.

1. Go to the **UnitOps** tab in the workbook and press the ***Add UnitOp*** button.
2. From the available list select **PIPESYS extension** and click ***Add***.

3. On the **Connections** tab complete the form as shown in Figure 4.2.

Figure 4.2



4.3 Defining the Elevation Profile

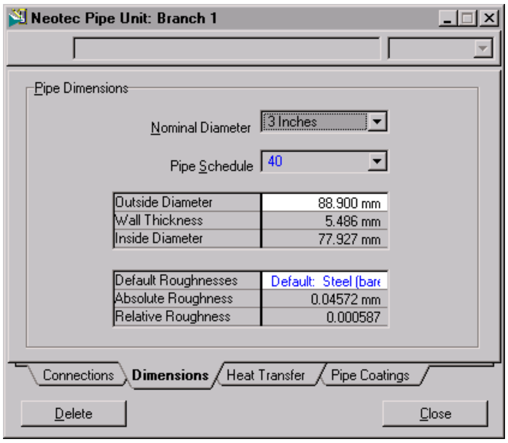
1. Open the **Elevation Profile** tab. As you can see from Figure 4.1, the coordinates of the Pipeline Origin have the value 0.0. Enter **0.0** into both the **Distance** and the **Elevation** cells in the **Pipeline Origin** group box.

Add a Pipe Unit to the matrix as follows:

2. First, select the **<empty>** cell in the **Pipeline Unit** column and then choose **Pipe** from the drop-down list on the Menu Bar. A Pipe Unit Property View will appear.

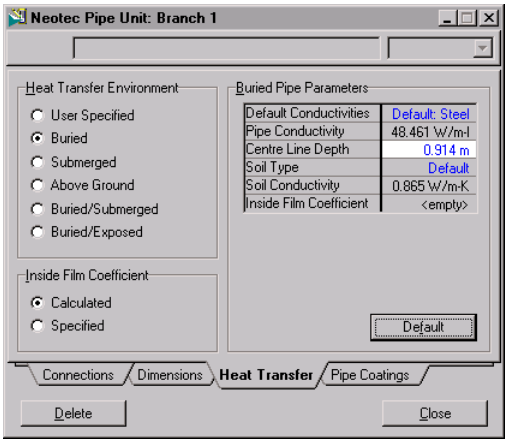
3. Complete the **Dimensions** tab of the Pipe Unit view by specifying a **Nominal Diameter** of **3 Inches** and a **Pipe Schedule** of **40**. Figure 4.3 shows the completed tab.

Figure 4.3



4. Go to the **Heat Transfer** tab of the Pipe Unit view. Select the cell that reads **<empty>** for the **Centre Line Depth** and the click the ***Default*** button. Figure 4.4 shows the completed tab.

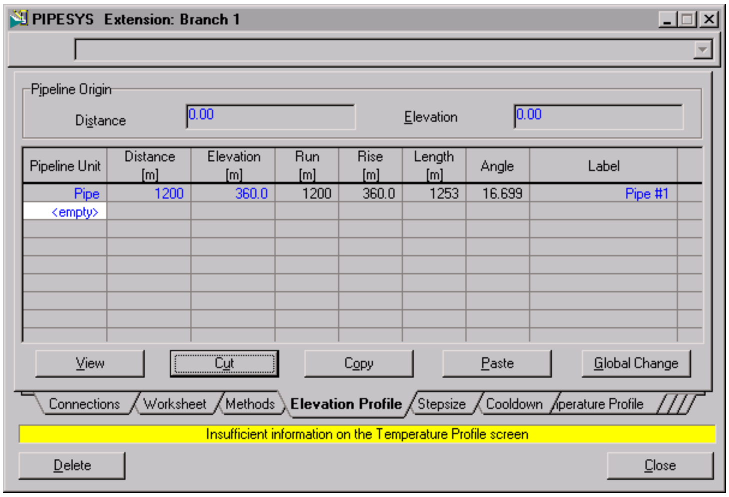
Figure 4.4



5. Close the complete Pipe Unit view.

6. The pipe unit will now appear as an entry in the matrix, with **<empty>** in all parameter cells. Pipe #1 has endpoint coordinates of (1200, 360). To complete the profile data entry, enter **1200** into the **Distance** cell and **360** into the **Elevation** cell. PIPESYS automatically calculates all the other parameters, as shown below.

Figure 4.5



7. Now add the second pipe unit to the matrix. Fill in the pipe unit view with the same specifications as were used for Pipe Unit #1. You may either re-enter all this information, or use the ***Copy*** and ***Paste*** buttons on the Elevation Profile tab.

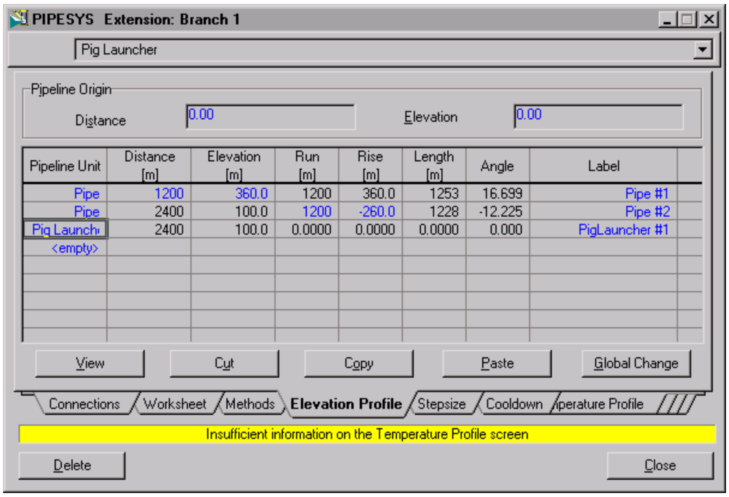
8. This time specify the second pipe unit endpoint using the **Run** and **Length** parameters instead of Elevation and Distance. Figure 4.1 shows that the second pipe unit has a **Run** of **1200** and a **Length** of **1227.84**. Enter these values into the Elevation Profile tab.

You may have noticed that the data on the Elevation Profile tab does not correctly represent the actual geometry of the pipeline. This is because *PIPESYS always assumes a positive angle for the pipe unit when the Run and Length parameters are used to specify the coordinates of the endpoint.*

9. To correct the matrix data, make a note of the **Angle** value, which is 12.23, and then delete the value in the Length cell. Now enter **-12.23** into the **Angle** cell. Or alternately, you could enter the value for the **Rise** as **-260** m.

10. To add the Pig Launcher, select the **<empty>** cell and choose **Pig Launcher** from the Edit Bar.

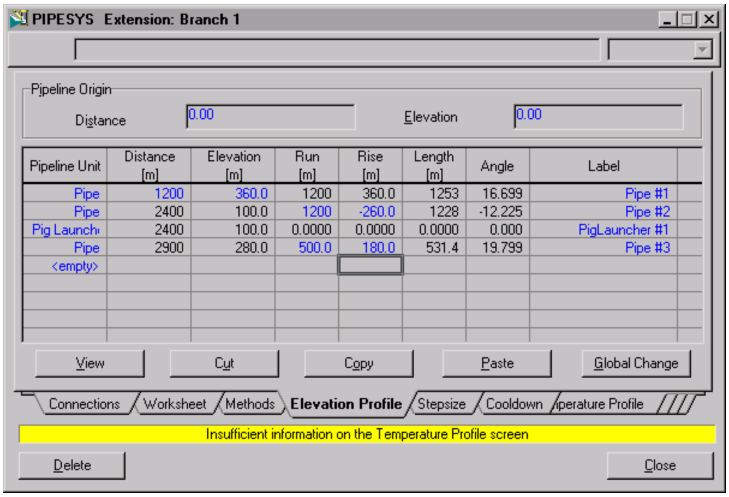
Figure 4.6



You are not required to specify any additional data to incorporate the Pig Launcher into the matrix. Figure 4.6 shows the **Elevation Profile** tab after the Pig Launcher has been added. Position data for the launcher or any other in-line facility does not have to be specified because this information is obtained automatically from the preceding component.

11. Finally, add a third pipe unit with the same parameters as the previous two. Using the **Run** and **Rise** parameters specify the endpoint coordinates. The **Run** value is **500** (2900-2400) and the **Rise** is **180** (280-100). Figure 4.7 shows the completed Elevation Profile tab.

Figure 4.7



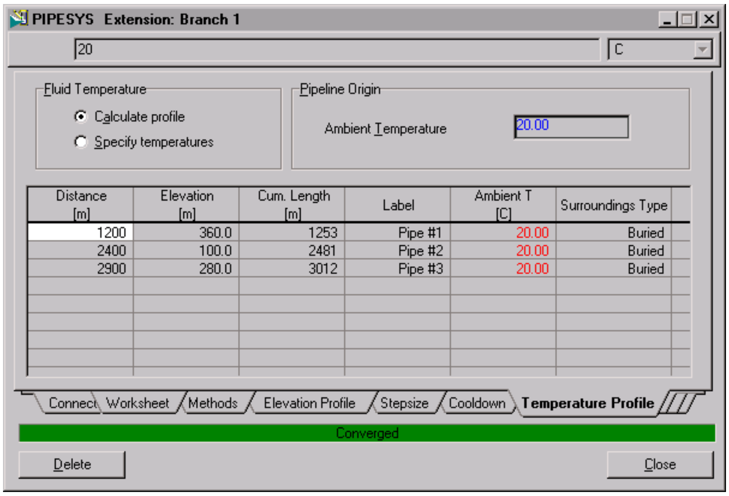
The status bar at the bottom of the PIPESYS view indicates that there is “Insufficient information on the Temperature Profile screen.”

12. Open the **Temperature Profile** tab. Enter **20** into the **Ambient Temperature** cell of the **Pipeline Origin** group box.

You will notice that the Ambient Temperature value is automatically copied in the **Ambient T** cell for each individual pipe unit, unless otherwise specified.

Once the Ambient Temperature information is provided, PIPESYS begins calculating. When completed, the status bar reads Converged. The **Temperature Profile** tab of the converged extension is shown in Figure 4.8 below.

Figure 4.8

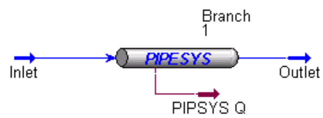


13. Save your completed case as Pipesys1.hsc.

The PFD generated for the completed case, plus a material stream table is shown below:

*To add a table to a PFD, right click on the PFD and choose **Add Workbook Table** from the drop down list.*

Figure 4.9



Material Streams			
		Inlet	Outlet
Vapour Fraction		1.0000	1.0000
Temperature	C	45.00	27.06
Pressure	kPa	8000	7183
Molar Flow	kgmole/h	300.0	300.0
Mass Flow	kg/h	6504	6504
Liquid Volume Flow	m3/h	17.78	17.78
Heat Flow	kJ/h	-2.775e+007	-2.799e+007

5 Pipe Unit View

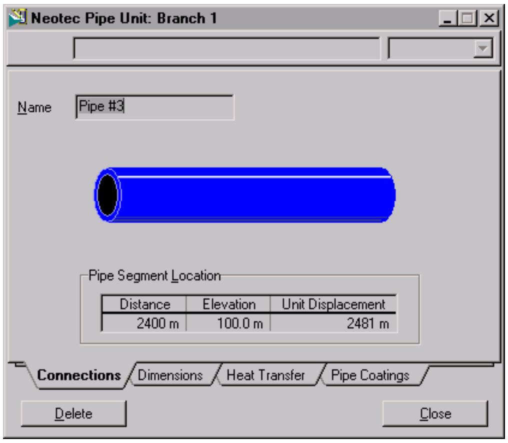
5.1 Connections Tab 3

5.1.1 Dimensions Tab 3
5.1.2 Heat Transfer Tab 5
5.1.3 Pipe Coatings Tab 8

5.2 Adding a Pipe Unit 9

This view is used to enter all parameters associated with the specification of a **Pipe Unit** in PIPESYS. All data settings related to physical characteristics, such as dimensions, roughness and coatings are entered here. This view also allows you to specify one of a number of external environments that affect the heat transfer from the flowing fluid, including below ground, open air and under water settings.

Figure 5.1



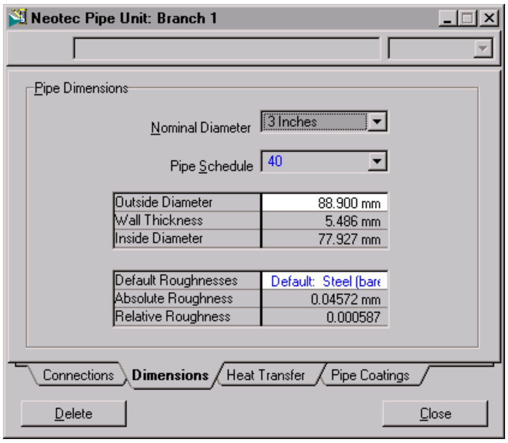
5.1 Connections Tab

Some basic information about the Pipe Unit is displayed on this tab. The pipe unit name and its profile location data appear here. The location data is repeated from the Elevation Profile tab of the Main PIPESYS View and is *read-only* here. If you wish to change the Distance, Elevation, or Unit Displacement data, you must return to the **Main PIPESYS View** and go to the **Elevation Profile** tab.

5.1.1 Dimensions Tab

The **Dimensions** tab features a built-in data set with a comprehensive range of pipe sizes and wall thicknesses. If you are using a standard pipe size in your project, you need only select a nominal diameter and a pipe schedule and PIPESYS will automatically fill in the other input cells. You can also use non-standard pipe sizes by manually entering all relevant data.

Figure 5.2

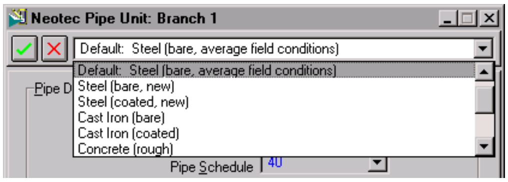


The Pipe Dimensions group box contains the following parameter input cells:

- **Nominal Diameter -** The commercial sizing descriptor for a given pipe size.
- **Pipe Schedule -** This drop-down box allows you to select from the American Standard B36.10 pipe wall thickness schedule or use the traditional standard weight (S), extra strong (XS) and double extra strong (XXS) specification method for entering the pipe nominal wall thickness value.
- **Outside Diameter -** A value will be automatically generated and entered here once a nominal diameter is selected. If you are dealing with a non-standard pipe size you can enter this value manually.
- **Wall Thickness -** The actual thickness of the pipe wall. Can be set manually.
- **Inside Diameter -** The actual inside diameter of the pipe. Can be set manually.
- **Default Roughness -** Wall roughness can be set by PIPESYS according to the pipe material entered in this input cell. If you have a specific value for roughness that you want to use instead, choose the User specified setting for Default Roughness. You will now be able to enter any value into the Absolute Roughness input cell.

Specify the Default Roughness by selecting from the list of materials in the drop-down bar.

Figure 5.3



- **Absolute Roughness -** The standard sand particle equivalent roughness rating used to define the effective roughness of the pipe. Pipe material, service time and environmental conditions can be factors in the determination of this value. PIPESYS has a comprehensive data-set of roughness values cross-referenced to pipe material types. Once you have chosen a pipe material, a corresponding roughness value will appear in this input cell. This parameter can be adjusted to match measured frictional pressure losses in existing pipelines.
- **Relative Roughness -** This value is calculated as the ratio of absolute roughness to inside pipe diameter.

5.1.2 Heat Transfer Tab

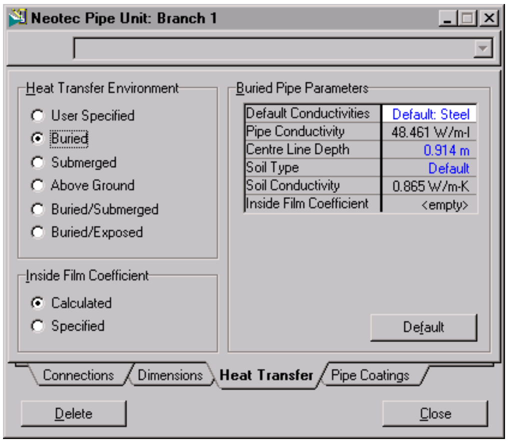
On this tab, a number of different heat transfer environments can be specified and the parameters that influence the rate of heat transfer from the flowing fluid specified. Figure 5.4 shows the **Heat Transfer** tab for the Pipe Unit view.

The following environments are available in the **Heat Transfer Environment** group box:

- **User Specified -** If special circumstances preclude selection of any of the other environments or you wish to run your calculations using a specific value for the heat transfer coefficient rather than have PIPESYS calculate it for you, choose this setting and enter a value in the Overall Heat Transfer Coefficient input cell.
- **Buried -** If the pipe unit is completely below ground, choose this setting.
- **Submerged -** Used for pipe units that are completely immersed in water.
- **Above Ground -** Choose this setting if the pipe unit is completely above ground and surrounded by air.
- **Buried/Submerged -** Used for pipe units that are partly below ground and partly underwater.

- **Buried/Exposed -** Choose this setting if the pipe unit is partly below ground and partly exposed to air.

Figure 5.4



The **Inside Film Coefficient** group box has a setting that allows you to control how PIPESYS accounts for the effects of the inside film on heat transfer. The term “inside film” refers to the laminar sublayer that exists adjacent to the pipe wall. Heat transfer through this film is primarily by conduction, but the thickness of the film depends on the flow rate and the fluid properties. It is usual to define the resistance to heat transfer in terms of a convective coefficient. The inside film can have a significant influence on the heat flow and can account for as much as half of the overall heat transfer coefficient value. You may select **Calculated** and have PIPESYS calculate the inside film coefficient using fluid property data, or select **Specified** and enter the value yourself.

The **Parameters** group box, on the right half of the **Heat Transfer** tab, contains a list of environment parameters specific to the heat transfer chosen. The following list describes the parameters for the various environments. For dual environments, both sets of parameters will be available.

Common to All Pipe Environments

- **Default Conductivities -** This parameter is similar to the Default Roughness parameter of the Dimensions tab. The pipe material type determines the value of the Pipe Conductivity parameter, which is set automatically once the pipe material is

chosen. If you want to supply your own value for Pipe Conductivity, set Pipe Material to User Specified. The Pipe Conductivity input cell will become user-modifiable.

- **Pipe Conductivity -** This is the thermal conductivity of the specified pipe material.

Buried

- **Centre Line Depth -** The burial depth of the pipeline, measured from the ground surface to the centre line of the pipe.
- **Soil Type -** You may select from a variety of commonly encountered soil types or choose User Specified. The soil type is used by the program to determine a value for the soil conductivity. If you have chosen User Specified, you may enter your own value in the Soil Conductivity input cell.
- **Soil Conductivity -** The thermal conductivity of the soil surrounding the pipe.

Submerged

- **Water Density -** The density of the water surrounding the pipe.
- **Water Viscosity -** The viscosity of the water surrounding the pipe.
- **Water Conductivity -** The thermal conductivity of the water surrounding the pipe.
- **Water Velocity -** The cross pipe velocity of the water surrounding the pipe. This value is used in convective heat transfer calculations.
- **Water Heat Capacity -** The specific heat capacity of the water surrounding the pipe.

Above Ground

- **Air Density -** The density of the air surrounding the pipe.
- **Air Viscosity -** The viscosity of the air surrounding the pipe.
- **Air Conductivity -** The thermal conductivity of the air surrounding the pipe.
- **Air Velocity -** The cross-pipe velocity of the air surrounding the pipe unit. This value is used in convective heat transfer calculations.
- **Buried Fraction -** The fraction of the pipe diameter that is underground. This number must be a value between 0.0 and 1.0.
- **Inside Film Coefficient -** Displays the calculated or user-entered value for the inside film coefficient.

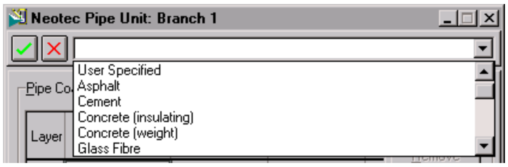
If you want PIPESYS to supply a default value for any of the Parameters data, highlight the input cell and press the ***Default*** button in the lower

right corner of the group box. PIPESYS will supply a default value to the input cell.

5.1.3 Pipe Coatings Tab

If the pipe has insulating and protective coatings, the relevant data can be entered into the matrix on this tab. You should begin with the innermost coating for Layer 1 and proceed outwards. To enter the data for a coating layer, select the cell in the **Coating** column containing **<empty>**. From the drop down input cell at the top of the tab (see Figure 5.5), you can then choose from a number of coating types.

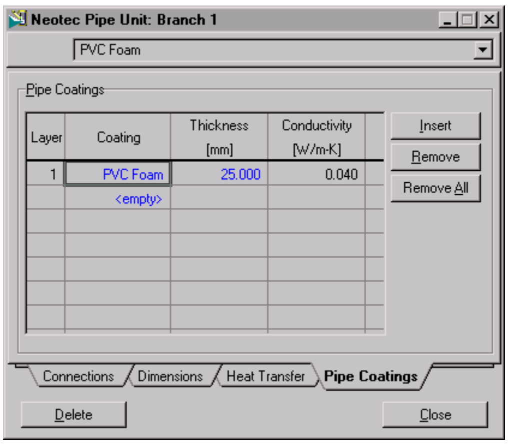
Figure 5.5



Once a coating type has been selected, the corresponding conductivity value for that material will appear in the **Conductivity** column. Complete the layer description by entering a value for the thickness.

If you want to add a new entry at an intermediate point on the list, select a cell in the row that will follow the position of the new entry. Press the ***Insert*** button and an empty row will be created for you to enter data. The ***Remove*** and ***Remove All*** buttons are used respectively to delete a particular row and to delete the entire matrix.

Figure 5.6

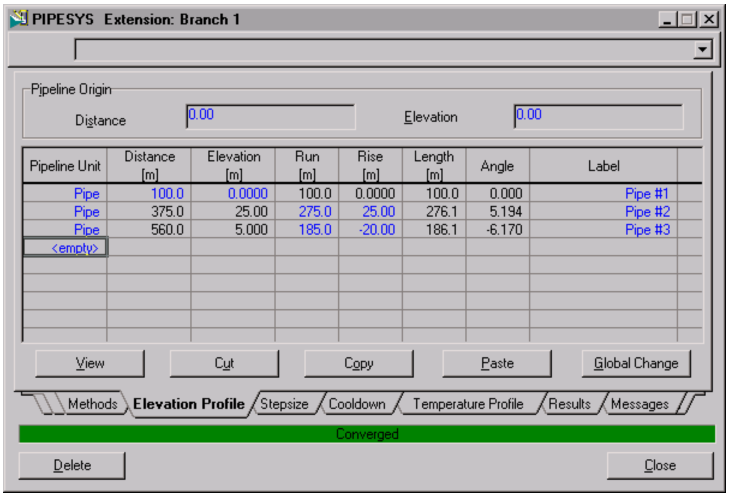


5.2 Adding a Pipe Unit

Carry out the following steps to define the pipe units:

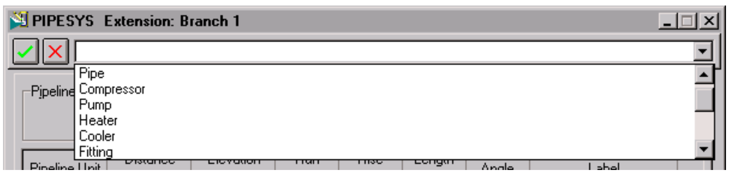
1. Open the Elevation Profile tab of the Main PIPESYS View. If the table is not empty you may add the Pipe Unit to the end of the component list or insert it between two components already in the list.

Figure 5.7



To **add** a new Pipe Unit at the end of the pipeline, select the cell containing **<empty>** in the **Pipeline Unit** column. Select **Pipe** from the Edit Bar drop down list. If you want to **insert** a new Pipe Unit within a set of Pipe Units, select the Pipeline Unit that will be immediately downstream of the new Pipe Unit and choose **Pipe** from the Edit Bar drop down list.

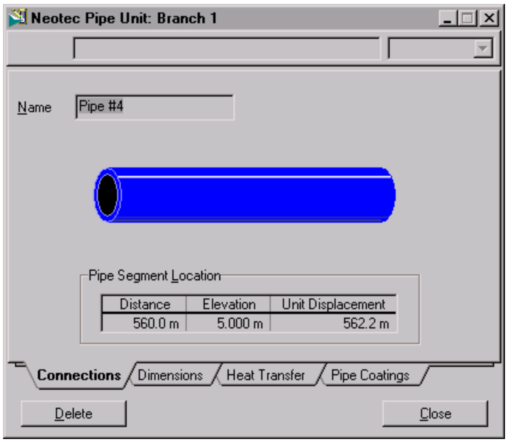
Figure 5.8



In both cases, a new **Pipe Unit** will appear in the profile matrix and the Pipe Unit View will open.

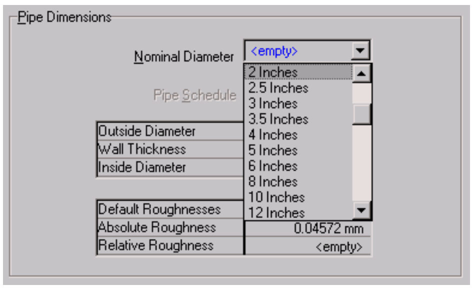
The Pipe Segment Location data on this tab is read-only. It can be changed only on the Elevation Profile tab of the Main PIPESYS view.

Figure 5.9



2. Enter a label for this Pipe Unit or accept the default name provided automatically in the **Name** cell.
3. Select the **Dimensions** tab on the Pipe Unit View. Here you enter the physical dimensions and the effective roughness of the pipe. If the nominal diameter and the pipe schedule are known, choose these settings from the **Nominal Diameter** and **Pipe Schedule** drop down boxes.

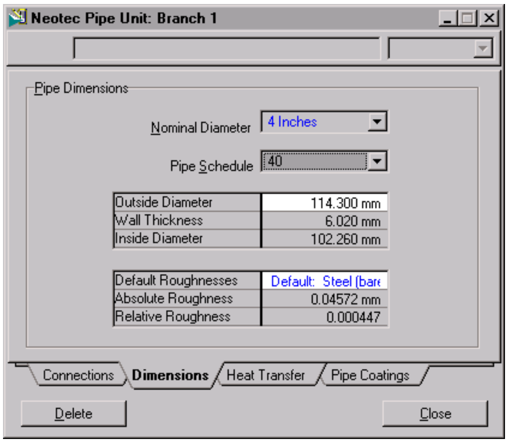
Figure 5.10



The program will obtain the corresponding dimensions from its internal database and fill in the **Outside Diameter**, **Wall Thickness** and **Inside Diameter** cells. Alternatively, you can select **User Specified** in the **Nominal Diameter** drop down box and enter these values directly.

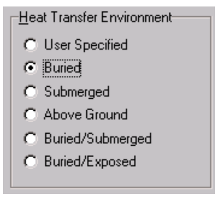
4. Choose a material type from the **Default Roughness** drop down list or enter an **Absolute Roughness** to complete the **Dimensions** tab. PIPESYS will use a default value for the roughness based on the material type that you select, or, if you choose **User Specified** for the **Material Type**, you will be able to enter a specified roughness value.

Figure 5.11



5. On the **Heat Transfer** tab, select the pipe surroundings for your case in the **Heat Transfer Environment** group box.

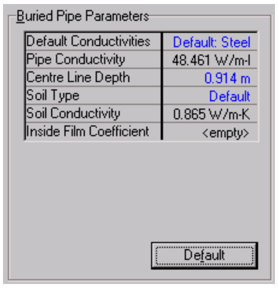
Figure 5.12



PIPESYS can calculate the heat transfer to the surroundings based on the characteristics of one of the external environments: **Buried**, **Submerged**, **Above Ground**, **Buried/Submerged** or **Buried/ Exposed**. A matrix of required parameters, as in Figure 5.13, will appear in the group box on the right of the form when Heat Transfer environment is chosen. When **User Specified** is selected, an overall heat transfer coefficient for the system may be entered.

See ***Section 5.1.2 - Heat Transfer Tab*** *for the definitions of the pipe environment parameters.*

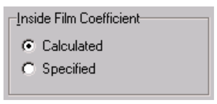
Figure 5.13



PIPESYS requires sufficient data to calculate the heat transfer from the fluid to the surroundings. Use the ***Default*** button to fill in required values for which you have no field data.

6. In the **Inside Film Coefficient** group box, select **Specified** to enter the resistance to heat transfer through the fluid film on the inside wall of the pipe. Select **Calculated** to have PIPESYS calculate the value for you. The default value is representative for turbulent flow.

Figure 5.14



This completes the information required for adding a pipe-unit.

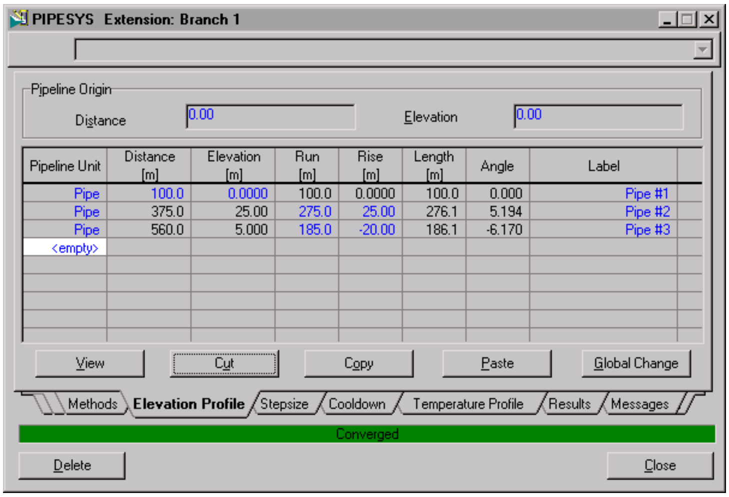
6 Global Change Feature

6.1 Global Change View **4**
6.1.1 Connections Tab 4
6.1.2 Dimensions Tab 5
6.1.3 Heat Transfer Tab 6
6.1.4 Pipe Coatings 6
6.2 Global Change Procedure **7**
6.3 Making a Global Change **9**

The **Global Change** feature is a convenient way to change the parameters of multiple **Pipe Units** in the elevation profile. A **Global Change** operation makes a simultaneous access to any or all of the Pipe Units in the elevation profile and changes a selection of parameters to your specifications. The **Global Change** feature has been implemented in PIPESYS as a time saving mechanism so that you are not required to open the property views for each individual Pipe Unit in order to make a change common to all units.

The ***Global Change*** button is accessed through the Elevation Profile tab of the Main PIPESYS view.

Figure 6.1



It can be used to edit the property view parameters for a single Pipe Unit and to subsequently duplicate the edits for none, some, or all of the other **Pipe Units** in the pipeline, in a single sequence of operations. This saves time when implementing changes to many Pipe Units at once. Of greater importance is that this feature reduces the potential for errors during the edit process. Any Pipe Unit can be used as a data template for changing the other Pipe Units in the pipeline simply by selecting it prior to clicking the ***Global Change*** button.

Figure 6.2

For example, after having made a pressure drop calculation for a pipeline consisting of 10 sections of 4" pipe, you might wish to repeat the calculation for the same pipeline with all diameters increased to 6". Rather than changing each of the 10 Pipe Units individually, you can apply the **Global Change** feature. Using this feature, you are required to execute only a few user-interface operations to change the pipe diameters from 4" to 6" for all **Pipe Units**.

The **Global Change Property View** is almost identical to the **Pipe Unit Property View**. Except as noted in this chapter, you make changes to Pipe Unit parameters using the same interface features that are described **Chapter 5 - Pipe Unit View**.

The distinguishing feature of the Global Change Property View is the ***Change*** check boxes that are associated with particular groups of parameters on each of the tabs, as shown in Figure 6.3.

Figure 6.3

These check boxes have two functions. They become automatically checked when you change one or more of their associated parameters as a reminder that you have requested a Global Data change for the selected Pipe Unit or other Pipe Units. As well, they can be manually checked to indicate that the data for the selected Pipe Unit has not changed but will be copied to other Pipe Units in the elevation profile.

6.1 Global Change View

6.1.1 Connections Tab

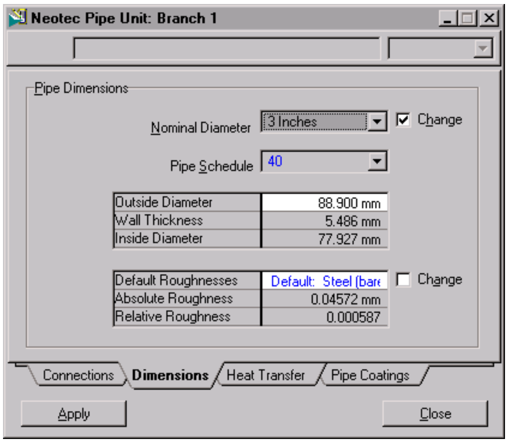
Displayed in the **Name** cell on this tab is the name of the Pipe Unit that was selected for the Global Change appended to the words "Copy of". This serves as a reminder that you are only working with a copy of the **Pipe Unit** data. No changes will be made to the original data until you first press the ***Apply*** button, select some **Pipe Units** to change and then finally press the ***Close*** button to close the Global Change View. This will then initiate the recalculation of the PIPESYS extension.

Figure 6.4

6.1.2 Dimensions Tab

This tab is identical to the **Dimensions** tab of the Pipe Unit Property View except for the **Change** check boxes beside the **Nominal Diameter** cell and the **Roughness Data** matrix.

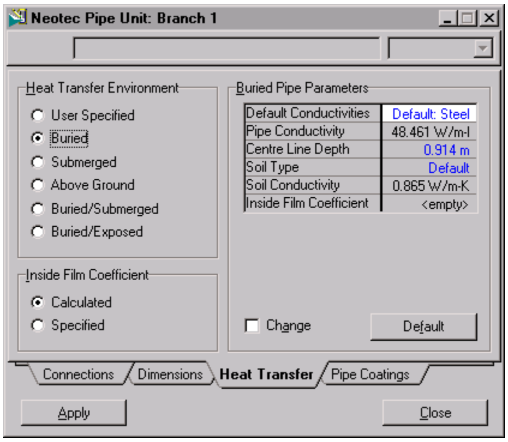
Figure 6.5



6.1.3 Heat Transfer Tab

On this tab you can change any of the parameters that affect the heat transfer from the fluid system. The type of environment, method of inside film coefficient derivation and the parameters associated with the environment can all be altered during a Global Change operation.

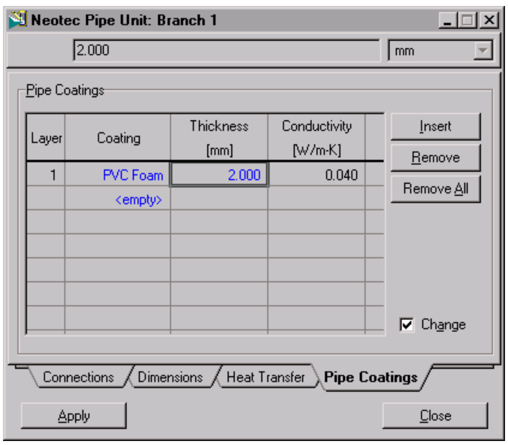
Figure 6.6



6.1.4 Pipe Coatings

The Pipe Coatings tab lists in matrix form the insulating coatings applied to the Pipe Unit.

Figure 6.7



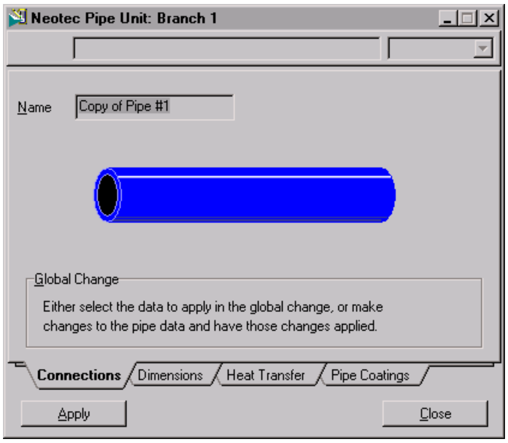
6.2 Global Change Procedure

The steps to implementing a global parameter change for some or all of the Pipe Units in the elevation profile are outlined in the following procedure:

1. Select any one of the **Pipe Units** in your elevation profile matrix and press the ***Global Change*** button. The **Global Change Property View** will appear and display the data from the selected unit.

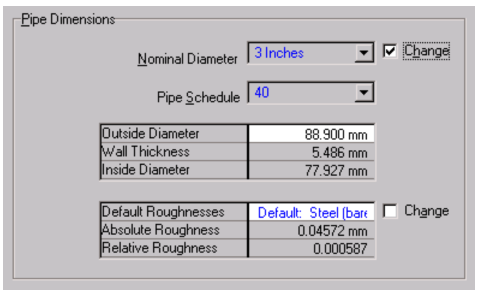
The Global Change Property View is almost identical to the Pipe Unit Property View.

Figure 6.8



2. Request a **Global Change** for a particular parameter by entering the new parameter values into the input cells. Each major group of Pipe Unit parameters has a check box beside it which will become automatically checked once a parameter has been changed.

Figure 6.9

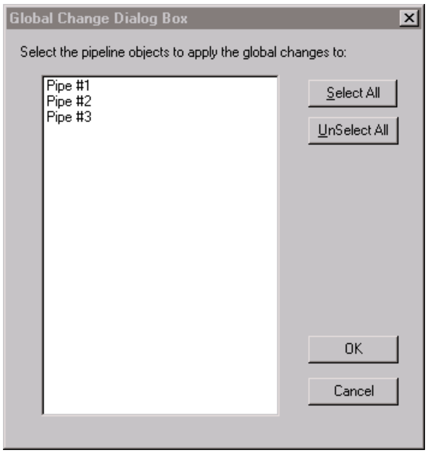


The check box also allows you to specify that the data of a particular unit, as it appears, will be duplicated to other Pipe Units in the elevation profile. To do this, just click on the check box beside each parameter that you want copied to other Pipe Units but leave the data unchanged.

3. Once all the changes that you want to make have been specified, press the ***Apply*** button and the **Global Change Dialogue Box** will appear with a list of all the Pipe Units in the profile.

To select more than one individual Pipe Unit, hold down <shift> while selecting the desired units.

Figure 6.10



Select the Pipe Units in the list that will be subjected to the Global Change and press the ***OK*** button.

4. To complete the **Global Change**, you must close the view by pressing the ***Close*** button. PIPESYS will then make the requested parameter changes to all Pipe Units selected for the Global Change procedure.

6.3 Making a Global Change

Example

The **Global Change** feature has been implemented in PIPESYS as a time saving mechanism so that when making a change common to more than one pipe unit, you do not need to open each Property View and change the data manually. A **Global Change** operation makes a simultaneous access to any or all of the Pipe Units in the elevation profile and changes a selection of parameters to the desired values.

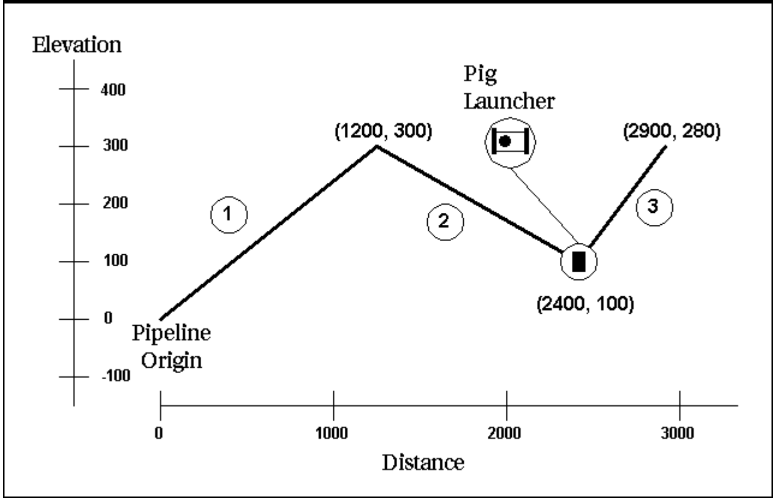
The **Global Change** feature can be used to edit the **Property View** parameters for a single **Pipe Unit** and to subsequently duplicate the edits for some or all of the other Pipe Units in the pipeline. Any Pipe Unit can be used as a data template for changing the other Pipe Units in the pipeline.

This short example uses the case **Pipesys1.hsc** that you created in **Chapter 4 - Elevation Profile -Quick Start**. This case consists of a single PIPESYS extension comprised of 3 segments of steel pipe and a pig launcher situated between the second and the third pipe units. The pipe is buried, has a 3” diameter and is schedule 40. If you have not yet completed this case, you must do so before proceeding with this example.

In this example, the Global Change feature will be used to change the diameter of all the pipe units from 3” to 4”. Figure 6.11 shows an

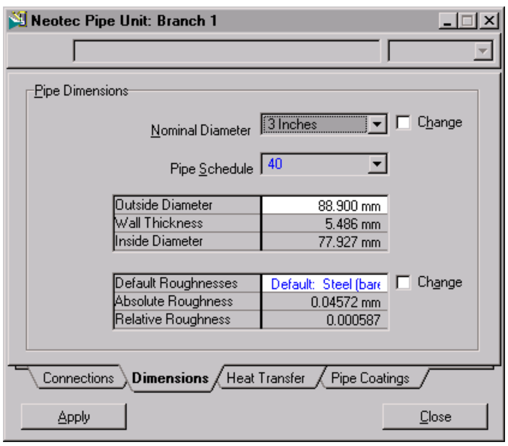
elevation profile diagram of the pipeline that is to be modified.

Figure 6.11



1. Highlight the Pipe Unit #1 in the **Pipeline Unit** column on the **Elevation Profile** tab and press the ***Global Change*** button.
2. When the **Global Change Pipe Unit View** appears, open the **Dimensions** tab as shown in Figure 6.12.

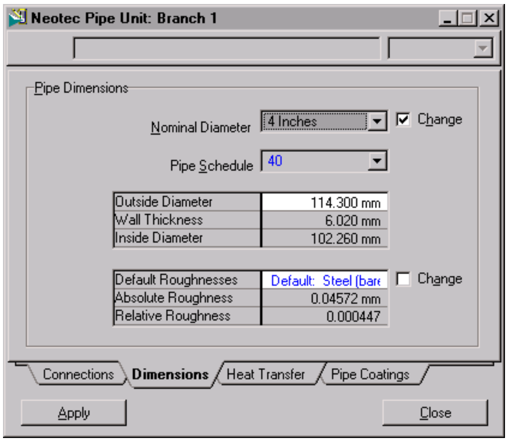
Figure 6.12



The check beside a Pipe Unit parameter indicates that it has been changed and that this change can subsequently be applied to other Pipe Units.

3. Now change the Pipe Unit diameter to 4". Select **4 Inches** from the **Nominal Diameter** drop down cell and you will see that a check appears in the **Change** check box beside it. You will also have to re-select **Schedule 40** from the Pipe Schedule drop down cell. Figure 6.13 shows the changed view.

Figure 6.13



4. Now press the ***Apply*** button at the bottom of the **Global Change View**.

5. The **Global Change Dialog Box** will appear. Here you must specify which Pipe Units in the pipeline will be subjected to the Global Change. In this case the change will apply to all pipe units, so press the ***Select All*** button to highlight all the pipe units. Figure 6.15 shows the dialog box with the selected pipe units.

Figure 6.14

The Global Change view must be closed to initialize the PIPESYS calculations.

6. Press the ***OK*** button on the **Global Change** view and PIPESYS will recalculate the extension using the new diameter value.

This completes the Global Change example. The following table compares calculated results for the **Outlet** stream for the 3” and 4” diameter pipeline.

Diameter	3”	4”
Vapour	1.00	1.00
Temperature [C]	27.06	28.06
Pressure [kPa]	7182.73	7630.47
Molar Flow [kgmole/h]	300.00	300.00
Mass Flow [kg/h]	6504.44	6504.44
Liq Vol Flow [m3/h]	17.78	17.78
Heat Flow [kj/h]	-2.79916e+07	-2.80155e+07

7 In-line Compressor

7.1 The Compressor View.. 3

7.1.1 Connections Tab 3
7.1.2 Parameters Tab... 4
7.1.3 Curve Tab.. 7
7.1.4 Fuel Requirements Tab... 9
7.1.5 Mechanical Losses Tab.. 10
7.1.6 Requirements Tab...11
7.1.7 Results Tab 12

7.2 Adding a Compressor.. 13

This view is used to enter all of the data required to specify the characteristics of a compressor unit in a PIPESYS extension. PIPESYS contains five different compressor models which operate either on performance data built into the program or on data entered by the user.

Compressors can be installed in pipelines in which there is also a liquid phase. If conditions are such that there is a liquid phase at the compressor suction side, PIPESYS will automatically remove the liquid and perform the compression calculations on the remaining gas phase. The liquid phase is assumed to be pumped around the compressor and the two phases are then recombined at the discharge side.

7.1 The Compressor View

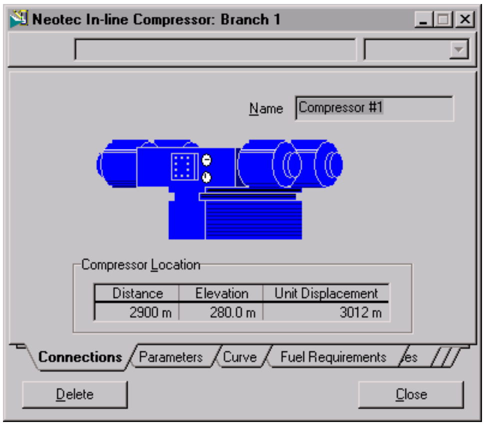
7.1.1 Connections Tab

You may give the compressor a unique name by entering a label into the **Name** input cell. This label is the same as that displayed in the elevation profile matrix in the Main PIPESYS View. The location of the compressor in the profile is specified in the Compressor Location group box using these parameters:

- **Distance -** the horizontal position of the compressor using the Pipeline Origin as the reference point
- **Elevation -** the vertical position of the compressor using the Pipeline Origin as the reference point
- **Unit Displacement -** the length of the true flow path from the pipeline origin to the compressor

The data displayed in the Compressor Location group box may not be edited. You must go to the Elevation Profile tab in the Main PIPESYS View if you need to change the compressor location data.

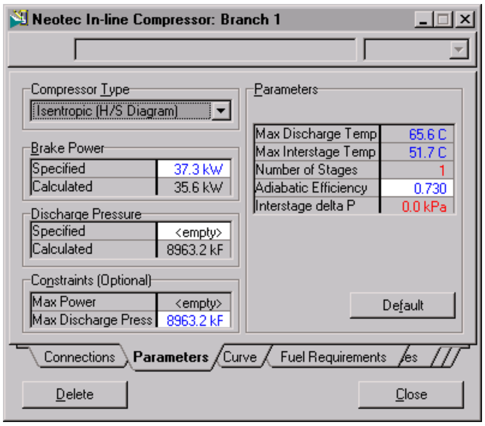
Figure 7.1



7.1.2 Parameters Tab

This tab is used to define the basic operating characteristics of the compressor. In the **Compressor Type** group box there is a drop down cell from which you can select one of five possible models for performing compressor calculations.

Figure 7.2



The five types of compressor models available in PIPESYS are:

- **Isentropic (H/S Diagram) -** A compressor that follows an adiabatic compression path as defined on an Enthalpy/Entropy diagram.
- **Polytropic (Internal Curve) -** The performance of this type is defined by a set of generalized curves contained within PIPESYS. These curves describe the relationship between the required brake horsepower per unit of volumetric flow (referenced to one atmosphere and the suction temperature) and the compression ratio, for gases with various specific heat capacity ratios.
- **Polytropic (User Curve) -** This compressor is identical to the Polytropic (Internal Curve) type except that the performance is defined by data which you must enter on the Curve tab.
- **Isentropic (GPSA) -** This compressor follows an adiabatic cycle such that PV^k = constant where P is pressure, V is specific volume and k is the heat capacity ratio Cp/Cv.
- **Polytropic (GPSA) -** This compressor follows an polytropic path such that PV^n = constant where n is called the polytropic exponent.

Selection of one of these types should be based on your knowledge of the specific compressor that you are modelling. Once a particular compressor type is selected, the tab will change accordingly and display input cells in which you can enter the values used to characterize the compressor. Each compressor type has some or all of the following parameters that must be entered:

Brake Power

- **Specified -** The Brake Power is the total power for all stages. You need to specify only one of Brake Power or Discharge Pressure and PIPESYS will calculate the other. Once you enter a value here, PIPESYS will calculate the corresponding discharge pressure and display it in the Discharge Pressure group box.
- **Calculated -** PIPESYS will display the brake power calculated from the specified Discharge Pressure in this input cell.

Discharge Pressure

- **Specified -** This value is the pressure at the outlet of a single or multistage compressor. You need to specify only one of Brake Power or Discharge Pressure and PIPESYS will calculate the other. Once you enter a value here, PIPESYS will calculate the corresponding brake power and display it in the Brake Power group box.

- **Calculated -** PIPESYS will display the discharge pressure calculated from the specified Brake Power in this input cell.

Optional Constraints

- **Max Power -** If the compressor discharge pressure has been specified, you can enter a value here to constrain the computed power requirement. If no value is entered here, PIPESYS will make its calculations with the assumption that the compressor is capable of supplying as much power as is needed to attain the specified discharge pressure. However, this can be an unrealistic assumption. The compressor may be incapable of such performance, in which case you can find the greatest discharge pressure that it can deliver without exceeding its rated maximum power. To do so, enter values for Discharge Pressure and Max Power. If PIPESYS finds that the compressor must exceed the Max Power setting to match the discharge pressure, it will recalculate and find the greatest discharge pressure that it can deliver at the maximum power setting.
- **Max Discharge Pressure -** If the compressor power has been specified in the Brake Power cell, you can constrain the compressor discharge pressure by entering a value in this cell. In this way, you can ensure that you do not exceed the maximum operating pressure for your pipeline. If the specified power will cause the compressor to exceed the maximum discharge pressure, then the compressor discharge will be set to this value and a new lower brake power will be computed.

Parameters

- **Max Discharge Temperature -** The temperature of the compressor discharge is limited to this value by cooling the gas. The theoretical duty for the cooler is reported on the Requirements tab.
- **Max Interstage Temperature -** This parameter is applicable only to multi-stage compressors. If the temperature of the gas at any of the intermediate discharges exceeds this value, PIPESYS will automatically install an interstage cooler to lower the temperature of the gas to this value. This duty is also reported on the Requirements tab.
- **Number of Stages -** You may specify any number of stages for a multi-stage compressor in this input cell. If you leave this empty, PIPESYS will compute the number of stages based on a maximum compression ratio of 4:1.
- **Polytropic Efficiency -** PIPESYS uses a default value of 0.73 unless you specify otherwise.
- **Adiabatic Efficiency -** PIPESYS uses a default value of 0.73 unless you specify otherwise.

- **Interstage deltaP -** This parameter corresponds to the pressure loss caused by the interstage tubing and fittings. PIPESYS uses a value of zero as a default value.
- **Heat Capacity Ratio -** This parameter is a property of the gas and is expressed as Cp/Cv. It is needed for temperature rise calculations and for selecting values from the built-in compressor performance curves. PIPESYS will calculate this value using gas property data as a default or will use whatever value you choose to enter.
- **Rating Factor -** This is a calibration factor which you can use to fine-tune a compressor performance curve for either internal or user-specified curves. This factor arbitrarily increases or decreases the power value obtained from a performance curve. Its purpose is to allow you to more closely model the performance of an actual compressor using the built in performance curves in PIPESYS.

7.1.3 Curve Tab

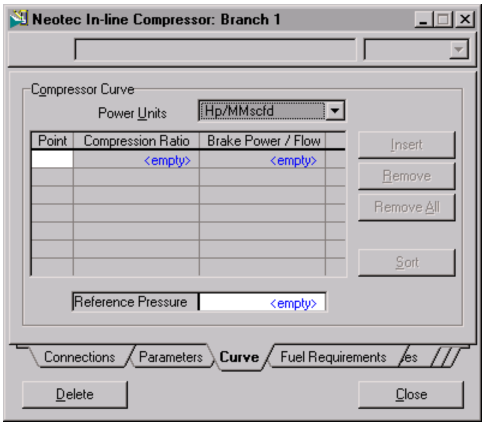
Use this tab to enter performance curve data when the Polytropic (User Curve) on the Parameters tab is chosen. Otherwise, the message “No data is required for the compressor type selected” will appear.

You must select the units used to specify the Brake Power/Flow parameter. You may choose from:

- **Hp/MMscfd -** Horsepower per million standard cubic feet per day
- **kW/E3sm3d -** kilowatts per thousand standard cubic meters per day

You must also enter the curve data into the matrix. You should be able to obtain this information from the compressor manufacturer’s specification sheets. You must enter at least two data points to completely specify the compressor curve. It is not necessary to enter the Point value as these numbers are automatically generated. You must enter a value into the **Reference Pressure** input cell. This value is the pressure at which the compressor was tested and should be recorded on the compressor specification sheets.

Figure 7.3

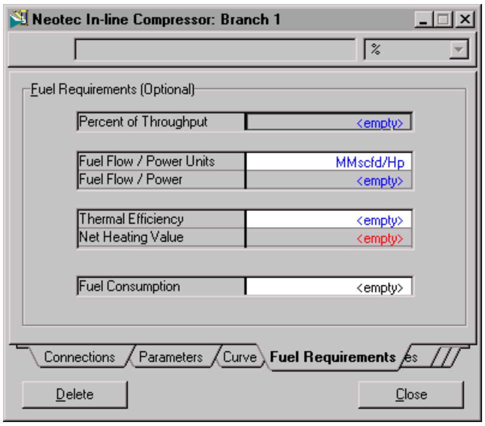


The ***Insert***, ***Remove*** and ***Remove All*** buttons can be used to alter the curve data after all points have been entered. To insert, select the entry that immediately follows the position where you want the new point to be located. Press the ***Insert*** button and blank data cells will appear in the list. To remove a data point, select the specific cell to be removed and press the ***Remove*** button. If you want to clear the list and start over, press ***Remove All***.

7.1.4 Fuel Requirements Tab

PIPESYS Compressor Units can remove gas from the stream being compressed to satisfy fuel gas requirements. Data required to compute the fuel gas requirements for the compressor is contained on the Fuel Requirements tab of the Compressor Pipeline Unit.

Figure 7.4



If fuel gas calculations have been requested, PIPESYS will compute the fuel gas requirements for the compressor based on:

- a specified **percentage of throughput**.
- a specified ratio of fuel gas to the overall brake horsepower.
- the net heating value of the gas, thermal efficiency and overall brake horsepower.

In all of the above cases, it is assumed that the fuel gas is taken from the suction side of the compressor after any first stage separation is done to remove liquids. Fuel consumption reduces the total amount of gas that must be compressed and thus, an iterative solution is applied to compute the compressor horsepower.

For the case when the net heating value of the gas and the overall brake horsepower is used to compute the fuel requirements, the thermal efficiency of the compressor is required. A value for the net heating value of the gas can also be entered. If the value of the net heating value is left empty, it is determined from the gas being compressed. The fuel requirements are then computed as,

$$Q_f = \frac{(0.706798)BHP}{h_v \cdot \eta_t}$$

where: Q_f = total gas removed from the pipeline at the compressor station (lb/sec)

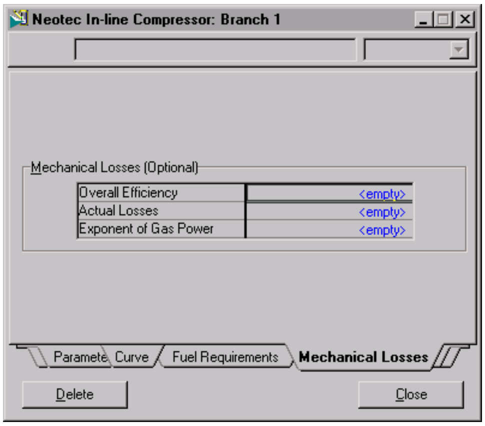
h_v = net heating value of the gas (BTU/lb)

n_t = overall thermal conversion efficiency

7.1.5 Mechanical Losses Tab

There are three optional parameters that can be entered on this tab. These values improve the accuracy with which the compressor is modelled and it is recommended that you supply the program with this data if you are able to obtain it from the manufacturer's specification sheets. If you leave the input cells empty PIPESYS will ignore mechanical losses. Only one of these parameters may be set to a non-zero value as they represent different and mutually exclusive ways of describing the mechanical losses that occur in a compressor process.

Figure 7.5



The Power Balance for a compressor is:

Brake Power = Gas Power + Mechanical Losses

Brake Power is the total power required to operate the compressor. Gas Power is the sum of the theoretical power (the power needed to compress an ideal gas) and the additional power needed to compensate for compression losses.

Typically, mechanical losses amount to 1% to 3% of the total brake power. There are three ways of specifying the mechanical losses:

1. **Overall Efficiency** - This is a number less than one where:

   Mechanical Losses = (1 - Mechanical Efficiency) x Gas Power

2. **Actual Losses** - Expressed as an actual power value, in units consistent with the other compressor parameters.

3. **Exponent of Gas Power** - This value is used in the expression:

   Mechanical Losses = Gas Power ^{x}

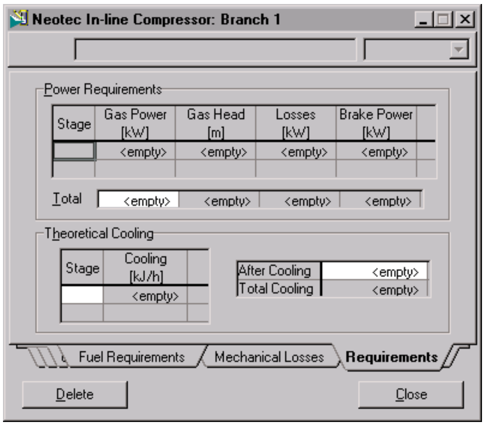
A typical value for the exponent x is 0.4.

7.1.6 Requirements Tab

Here the operational requirements for your compressor are displayed. These are the values calculated by PIPESYS and broken down on a per stage basis. The **Power Requirements** group box contains a summary of required Gas Power, Gas Head, Losses and Brake Power for each stage and a total of these values for the entire compressor.

The **Theoretical Cooling** group box displays any cooling that was required, if this proved to be necessary, for each stage and the compressor outlet. The sum of all duty required for cooling the gas is displayed in the Total Cooling cell.

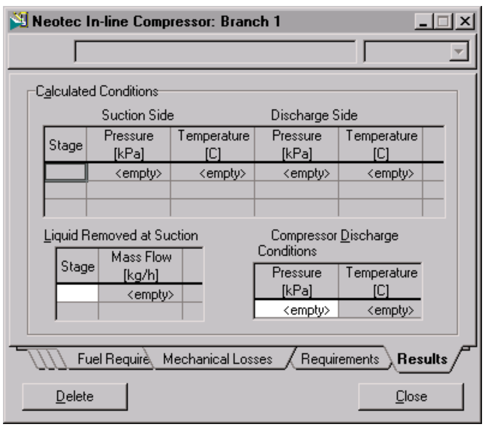
Figure 7.6



7.1.7 Results Tab

The conditions at the suction side and the discharge side for each stage of the compressor are displayed on this tab. See Figure 7.6. The compressor discharge conditions may be different from the conditions on the discharge side of the final stage if any after cooling has been installed. These conditions, for both pressure and temperature, are displayed in the **Compressor Discharge Conditions** group box.

Figure 7.7



If the inlet stream is multiphase either at the suction side of the compressor, or at the suction side of any subsequent stage, the liquid is removed and compression is computed on the basis of the resulting vapour phase. Any liquid removed is recombined with the outlet stream from the compressor, at the discharge pressure and temperature conditions. Furthermore, a warning is issued to alert the user that some liquid separation occurred prior to compression. In the **Liquid Removed at Suction** group box, the amount of liquid removed at each stage of the compressor is reported.

For the case when a stream is determined to be single phase liquid, either at the suction side of the compressor, or any subsequent stage, the entire stream is compressed and a warning:"Compressor - Single phase liquid encountered." is posted on the **Messages** tab. Thus the compressor is capable of handling dense phase fluids that are reported by the equation of state to be single phase liquid.

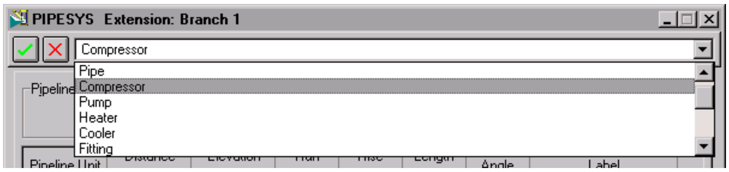
7.2 Adding a Compressor

Carry out the following steps to add a compressor to your pipeline:

To insert a compressor in the middle of the elevation profile, select the Pipeline Unit that will be placed immediately downstream and then add the compressor.

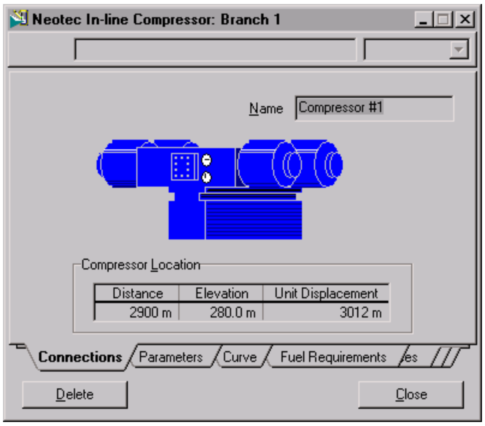
1. Open the **Elevation Profile** tab of the Main PIPESYS View. You can add the compressor to the end of the list of Pipeline Units or insert it at an intermediate point in the profile.
2. To add the new compressor at the end, select the cell containing **<empty>** in the **Pipeline Unit** column and choose **Compressor** from the Edit Bar at the top of the View.

Figure 7.8



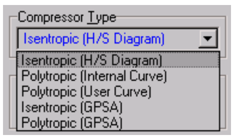
3. In the new compressor view that appears, enter a label in the **Name** cell on the **Connections** tab, if desired.

Figure 7.9



4. Open the **Parameters** tab on the Compressor View. Choose a compressor type from the drop down box in the **Compressor Type** group box. See Figure 7.9.

Figure 7.10



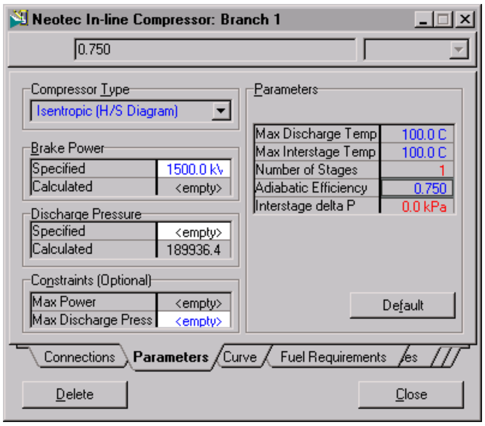
5. The compressor maximum output can be specified using one of two parameters. You can enter a value for the **Brake Power** or the **Discharge Pressure** in the **Specified** input cell of these group boxes. These are mutually exclusive parameters so entering a value for one precludes subsequently entering a value for the other.

Refer to ***Section 7.1.2 - Parameters Tab*** *for more detailed instructions regarding* ***Optional Constraints****.*

6. In order to constrain the compressor output, enter a value in the **Optional Constraints** group box. You can enter a value into the **Max Power** or the **Max Discharge Press** input cell depending on the compressor output method chosen in Step 4. Applying a

constraint is not required but you may find that applying a constraint allows you to find an optimal operating point for the compressor.

Figure 7.11

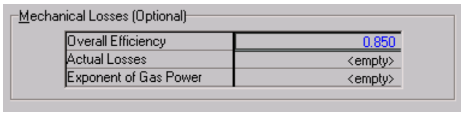


Parameter defaults are provided for:
- Adiabatic Efficiency
- Polytropic Efficiency
-Heat Capacity Ratio
- Rating Factor

7. Enter values in the **Parameters** group box that best represent the capabilities and type of your compressor. These values may be available from the manufacturer's specification sheets or may be known to you from previous experience with the compressor. Cells which will be filled in by the software but can be overwritten by you will be displayed in red. Optional values such as **Max Discharge Temp** and **Max Interstage Temp** can be left empty. Entries are required for all remaining cells. If any of the required parameters are unknown to you, pressing the ***Default*** button while the particular cell is selected will generate a default value.
8. Open the **Curve** tab. Depending on the type of compressor selected, you may have to supply data for a **Compression Ratio vs. Brake Power/Flow**. If no curve data is required, the message "No data required for compressor type selected" will appear on this tab. If curve data is required, a matrix will appear for data entry.

9. On the **Mechanical Losses** tab, there are three parameters used to specify mechanical losses for the compressor.

Figure 7.12

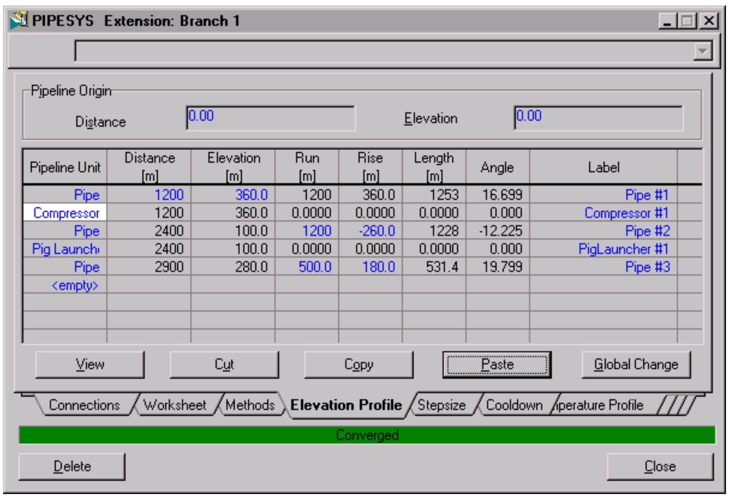


Only *one* of these may be selected as they are different and mutually exclusive methods for specifying mechanical losses. Entering a value for one of these parameters improves the accuracy with which the compressor is modelled and it is recommended that you supply the PIPESYS with this data if you are able to obtain it from the manufacturer's specification sheets. If you are unable to obtain this data, leave the input cells empty and the mechanical losses will not be computed.

10. At this point, you are finished entering the required data on the Compressor View. You can now close this View by pressing the ***Close*** button.

11. Finally, check that the location data for the compressor on the **Elevation Profile** tab of the Main PIPESYS View is correct. This data has been automatically determined using the position data of the preceding unit in the pipeline, so you should verify that it has been positioned as you intended. If not, you can use the ***Cut*** and ***Paste*** buttons on the Main PIPESYS View to transfer the compressor to a new position.

Figure 7.13



8 In-line Pump

8.1 In-line Pump View.. 3
8.1.1 Connections Tab 3
8.1.2 Parameters Tab.. 3
8.1.3 Curve Tab.. 5

PIPESYS supports two different methods of pump performance modelling. One method relates discharge pressure to fluid horsepower and volumetric flow rate using an equation. The other method uses tabular data that you must enter to establish the head and overall efficiency as functions of the volumetric flow rate. Both methods may be used interchangeably to model the performance of an actual pump so your choice of method is dictated by the type of data available.

Note: The pump can only be added to an ***all-liquid*** *system.*

8.1 In-line Pump View

8.1.1 Connections Tab

Figure 8.1

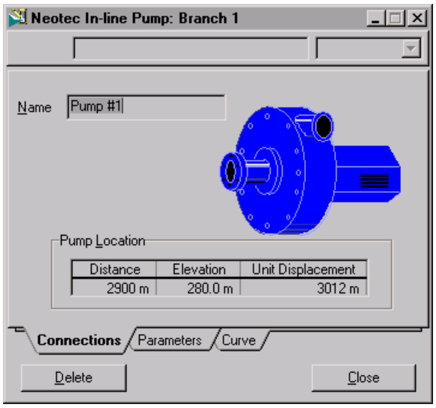


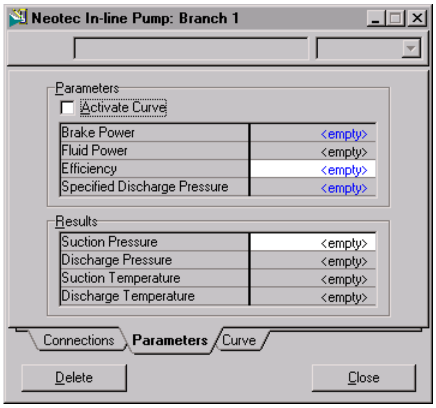
Figure 8.1 shows the **Connections** tab for the In-line Pump. The name and location of the pump are repeated here from the **Elevation Profile** tab of the Main PIPESYS View. The location, once defined in the Main PIPESYS View, cannot be changed in any other view and is displayed here only for your reference. If you need to change the location, open the **Elevation Profile** tab of the Main PIPESYS View.

8.1.2 Parameters Tab

Here you must choose to model the pump by using a built-in relation or by entering data to define a performance curve. If you check the **Activate Curve** check box, as shown in the upper left hand corner of Figure 8.2, the user-defined performance curve will be enabled and you are required to enter the curve data on the **Curve** tab. PIPESYS uses the

built-in relation when it is left unchecked. The only additional data you need to provide are values for some of the parameter input cells on this tab.

Figure 8.2



The parameters required to perform calculations with the built-in pump relation are **Brake Power** or **Specified Discharge Pressure** (these are mutually exclusive parameters) and **Efficiency**. The rest of the parameters on this tab are used to display results of the calculations. These parameters are defined as follows:

- **Brake Power -** The power required to operate the pump. Enter a value here or in the Specified Discharge Pressure input cell to provide enough data to perform the pump calculation.
- **Fluid Power -** The actual power delivered to the fluid system or the work done on the fluid per unit of time. The fluid power is related to the brake power by the relation:

$$\text{BHP} = \frac{FHP}{\eta}$$

where: BHP= Brake Horse Power,
FHP= Fluid Horse Power
η = pump efficiency

- **Efficiency -** The overall pump efficiency. A value of 0.70 is typical for some pumps, but you should consult the manufacturer's data sheet to obtain a value which properly represents the capabilities of your pump. This value is required by the program to calculate pump performance.

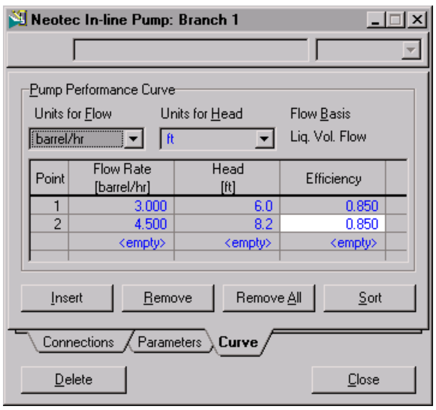
- **Specified Discharge Pressure -** The exit pressure from the pump. Either the Discharge Pressure or the Brake Power must be specified. If you have already specified Brake Power, you cannot supply a value for this parameter.
- **Suction Pressure -** Generally, the suction pressure, or pressure immediately upstream of the pump, is known from a calculation of the pressure to that particular location along the pipeline.
- **Discharge Pressure -** The calculated pressure immediately downstream of the pump.
- **Suction Temperature -** The temperature of the fluid at the pump inlet.
- **Discharge Temperature -** The temperature of the fluid at the pump outlet.

You are not required to enter any data values for these parameters if you are using the pump performance curve method. In fact, you will be unable to enter any parameter values on this tab if the **Activate Curve** check box is checked, as the parameter input cells will change to read-only mode.

8.1.3 Curve Tab

If the **Activate Curve** check box on the Parameters tab is checked, the **Pump Performance Curve** group box will appear, as displayed in Figure 8.3, and be ready for data entry on this tab.

Figure 8.3



Use the matrix on this tab to define the **Head vs. Flow Rate** and the **Efficiency vs. Flow Rate** curves for your pump. Curve data should be available from the pump manufacturer specification sheets. Different units for the **Flow** and **Head** data may be selected from the drop-down input cells above the matrix.

If you have *not* checked the **Activate Curve** check box on the Parameters tab, the message “No data is required as pump curve is not activated.” will be displayed on this tab.

9 In-line Facility Options

9.1 In-line Heater........ 3

9.1.1 Connections Tab 3
9.1.2 Parameters Tab........ 3

9.2 In-line Cooler 4

9.2.1 Connections Tab 5
9.2.2 Parameters Tab........ 5

9.3 Unit-X........ 6

9.3.1 Connections Tab 6
9.3.2 Parameters Tab........ 7
9.3.3 Results Tab 7

9.4 In-line Regulator........ 8

9.4.1 Connections Tab 8
9.4.2 Parameters Tab........ 8

9.5 In-line Fittings........ 9

9.5.1 Connections Tab 9
9.5.2 Parameters Tab........ 9

9.6 Pigging Slug Check........ 11

9.6.1 Connections Tab 12
9.6.2 Results Tab 12

9.7 Severe Slugging Check 13

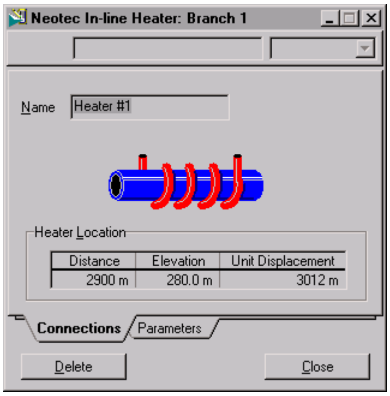
9.7.1 Connections Tab 15
9.7.2 Results Tab 15

9.8 Erosion Velocity Check 16
9.8.1 Connections Tab 17
9.8.2 Results Tab 18
9.9 Side Stream 18
9.9.1 Connection Tab 19
9.9.2 Parameters Tab 19

9.1 In-line Heater

The **In-Line Heater** can be used with any fluid system, but its effects are only considered for systems on which simultaneous pressure and temperature profile calculations are being performed. For example, the heater is ignored by the program if a user-specified temperature profile is entered because to have both a heater and a user-specified temperature profile over-specifies the system. PIPESYS copes with this situation by ignoring the effects of the heater.

Figure 9.1



9.1.1 Connections Tab

Figure 9.1 shows the **Connections** tab for the **In-line Heater**. As on all component views, the location for the unit is displayed as read-only data. If you need to change this data, open the **Main PIPESYS View** and go to the **Elevation Profile** tab.

9.1.2 Parameters Tab

On this tab you define the parameters that determine the effect of the heater on the fluid system. You must enter a value for the **Pressure Drop**, which is assumed to be constant and independent of the flow rate, and for one of either the **Temperature Rise**, **Specified Exit Temperature** or **Theoretical Duty**. The remainder of the cells are for displaying results.

The following parameters appear on this tab:

- **Pressure Drop -** The pressure loss experienced by any flow through the heater. A constant, flow rate invariant quantity.
- **Temperature Rise -** An incremental rise in temperature applied to the flow through the heater.
- **Specified Exit Temperature -** This is the required temperature for the fluids leaving the heater.
- **Theoretical Duty -** The amount of heat that must be transferred to the stream, based on the fluid properties and amounts of gas and liquid in the mixture, to achieve the required heating effect.
- **Inlet Temperature -** The temperature of the fluid at the heater inlet.
- **Exit Temperature -** The temperature of the fluid at the heater outlet.
- **Inlet Pressure -** The fluid pressure at the heater inlet.
- **Inlet Pressure -** The fluid pressure at the heater outlet.

If you have specified a fluid temperature profile for the pipeline, this tab will display the message; “Heater bypassed because you selected the option to specify the fluid temperature profile.”

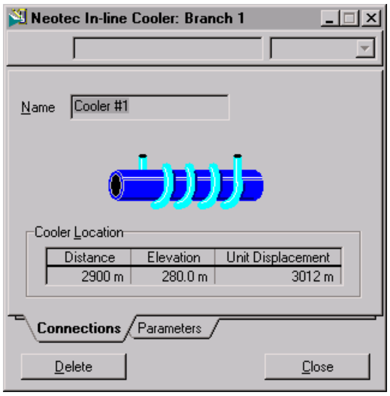
9.2 In-line Cooler

The **In-Line Cooler** is in all respects identical to the heater except that it removes heat from the flowing fluid instead of adding heat. Like the heater, the cooler has an effect on the fluid system only when both pressure and temperature profiles are being calculated.

9.2.1 Connections Tab

As on all component views, the location for the unit is displayed as read-only data. If you need to change this data, open the **Main PIPESYS View** and go to the **Elevation Profile** tab.

Figure 9.2



9.2.2 Parameters Tab

As with the heater, you must enter values for the **Pressure Drop**, which is assumed to be constant and independent of the flow rate, and for one of **Temperature Drop**, **Specified Exit Temperature** or **Theoretical Duty**. The remainder of the cells will display results based on the cooler calculations.

The following parameters appear on this tab:

- **Pressure Drop -** The pressure loss experienced by any flow through the cooler. A constant, flow-rate invariant quantity.
- **Temperature Drop -** An incremental drop in temperature applied to the flow through the cooler.
- **Specified Exit Temperature -** This is the required temperature for the fluids leaving the cooler.
- **Theoretical Duty -** The amount of heat that must be removed from the stream, based on the fluid properties and amounts of gas and liquid in the mixture, to achieve the required cooling effect.
- **Inlet Temperature -** The temperature of the fluid at the cooler inlet.
- **Exit Temperature -** The temperature of the fluid at the cooler outlet.

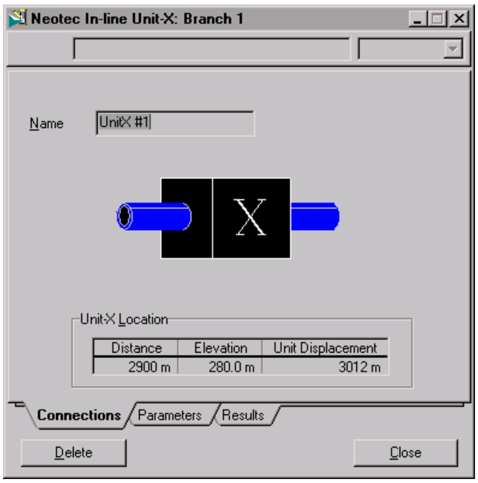
- **Inlet Pressure -** The fluid pressure at the cooler inlet.
- **Exit Pressure -** The fluid pressure at the cooler outlet.

If you have specified a fluid temperature profile for the pipeline, this tab will display the message; “Cooler bypassed because you selected the option to specify the fluid temperature profile”.

9.3 Unit-X

The **Unit-X** is a generic component that allows you to impose arbitrary changes in pressure and/or temperature on the fluid flow. It can be used to simulate the effects of a wide variety of process devices in a simple manner and is particularly useful in preliminary studies.

Figure 9.3



9.3.1 Connections Tab

As on all component views, the location for the unit is displayed as read-only data. If you need to change this data, open the **Main PIPESYS View** and go to the **Elevation Profile** tab.

9.3.2 Parameters Tab

Pressure parameters for the **Unit-X** can be specified in one of three ways on the **Pressure Parameters** group box. In the **Specified** column, you can enter a value for one of the following three parameters:

- **Pressure Change -** The difference in pressure between the inlet and outlet of the Unit-X is set to a fixed increment or decrement. Enter a positive value to specify a pressure gain and a negative value to specify a pressure drop.
- **Pressure Ratio -** Specifies a fixed value for the ratio expressed as the outlet pressure divided by the inlet pressure.
- **Exit Pressure -** This parameter allows you to specify a constant, absolute value of pressure at the unit outlet.

The **Calculated** column displays the actual, realized pressure parameters for the **Unit-X**. These may vary from the **Specified** values if the **Optional Constraints** parameters have been set in such a way as to limit the effect of the pressure parameter settings. For example, if you entered a Pressure Change of 100 kPa (a pressure increase of 100 kPa) and a Max Pressure Ratio of 1.5 for a Unit-X with an inlet pressure of 100 kPa, the outlet pressure would be constrained to 150 kPa rather than increase to 200 kPa.

Temperature parameters for the **Unit-X** can be set in one of two ways as follows:

- **Temperature Change -** Specifies a fixed increment or decrement for the fluid temperature across the unit. A negative value corresponds to a temperature decrease and a positive value to a temperature increase.
- **Exit Temperature -** Specifies a constant temperature value for the fluid at the unit outlet.

The parameters in the **Optional Constraints** group box allow you to limit the output of the Unit-X. The pressure ratio and exit pressure can be limited to whatever values you choose to enter in these input cells. Entering a value in the **Pressure Ratio** cell will cause the **Max Pressure Ratio** cell to be disabled and entering a value in the **Exit Pressure** cell will cause the **Max Exit Pressure** cell to be disabled.

9.3.3 Results Tab

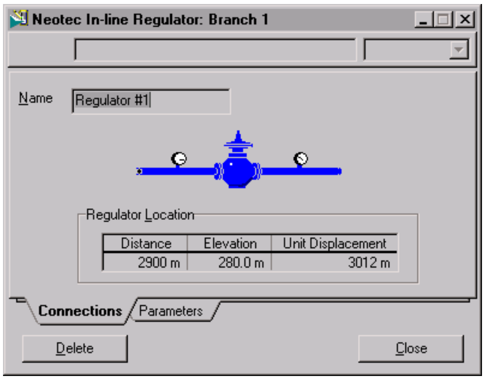
The inlet and exit conditions for the **Unit-X** are displayed here.

9.4 In-line Regulator

The **In-Line Regulator** is used to limit the pressure at any point in the pipeline profile. The only data required is the maximum exit pressure for the regulator. If the line pressure is less than the regulator exit pressure, *the regulator will be ignored*.

If the temperature profile is being calculated and the fluid is a gas-based system (i.e. dry gas, gas-water, gas-condensate), the discharge temperature will be computed assuming an isenthalpic expansion occurs. PIPESYS bases this computation on the specific enthalpy of the fluids and so the Joule-Thompson cooling effect is taken into account implicitly.

Figure 9.4



9.4.1 Connections Tab

As on all component views, the location for the unit is displayed as read-only data. If you need to change this data, open the **Main PIPESYS View** and go to the **Elevation Profile** tab.

9.4.2 Parameters Tab

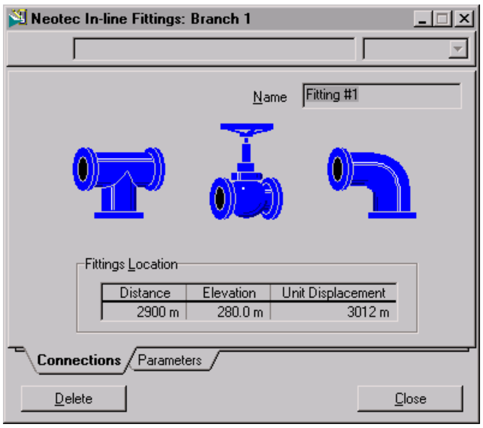
Enter the regulator discharge pressure into the **Maximum Exit Pressure** input cell. The **Results** group box displays the fluid pressure and temperature at the **Inlet** and **Exit** of the regulator once calculations are complete.

9.5 In-line Fittings

Fittings such as elbows, tees, valves and sudden expansions create a pressure drop in the system. For long pipelines or multiphase flow, it is rarely necessary to account for pressure losses through pipe fittings, unless there are a significant number of fittings. This is simply because the magnitude of the pressure drop caused by the fittings is small compared to the pressure losses brought about by all other causes.

There are a number of methods used to specify or calculate the pressure drop due to a fitting, but you may select any of these provided you have suitable data.

Figure 9.5



9.5.1 Connections Tab

As on all component views, the location for the unit is displayed as read-only data. If you need to change this data, open the **Main PIPESYS View** and go to the **Elevation Profile** tab.

9.5.2 Parameters Tab

In the **Fitting Selection** group box, you may select one of five methods for specifying the pressure loss across a fitting by specifying values for the following parameters:

- **Pressure Drop -** A constant value of pressure loss, independent of the flow rate.

- **Velocity Heads -** This method requires a value of the resistance coefficient or number of velocity heads, which you enter here, to calculate the pressure loss associated with the mixture velocity and density of the fluids.
- **Number of Diameters -** Characterizes the pressure loss as an equivalent length of pipe, measured in terms of pipe diameters, or the L/D ratio. The pressure loss across the fitting is calculated to be equal to that for a horizontal section of pipe having diameter D and length L.
- **Valve Coefficient -** This method uses the valve coefficient, which is defined as the flow rate of water, expressed in U.S. gallons per minute at 60^{o} F, that results in a pressure loss of 1.0 pounds per square inch across the valve when it is fully open. This coefficient is used in an expression for pressure loss that accounts for a variety of flow conditions and fluid properties. Although this expression was derived using water flow, it is still a useful guide for valves used in compressible flow (i.e. gas) service.
- **Hooper K1 & Hooper K2 -** These constants are used in the Hooper procedure to calculate the resistance coefficient, which is in turn used to calculate the pressure loss. The Hooper procedure is a reliable method for predicting the excess head loss in a fitting due to turbulence caused by abrupt changes in direction and speed of flow.

The **Inside Diameter** and **Absolute Roughness** group boxes are used to specify the inside diameter and inside surface roughness of the fitting. The first input cell in each group box is associated with a drop down list from which you may select **From Profile** or **User Specified**. If the former is selected, the program will obtain the diameter and roughness data from the adjacent components in the elevation profile. The **User Specified** setting allows you to enter your choice of values for diameter and roughness.

The **Results** group box displays the calculated results of the inlet and exit pressure and temperature for the fitting.

9.6 Pigging Slug Check

Pigging a pipeline to remove liquids or accretions of material on the inside pipe wall is a transient operation involving a time-varying buildup of a liquid slug in front of the pig.

Rigorous calculation of the growth of the slug is a highly complex procedure and is not attempted in PIPESYS, which is strictly a steady-state flow simulator. However, PIPESYS is capable of performing some simple calculations that are adequate for purposes of sizing a slug catcher.

Sizing calculations are made with the assumptions that there is no slippage of fluid past the pig and that the resident liquid fraction in the pipe is given by the steady state case. However, while the pig passes through the pipeline from the pig launcher to the pig trap, liquid continues to be produced from the downstream end at the steady state rate. When the pig finally reaches the trap, the amount of liquid in the slug is taken to be the total initial volume of liquid in the pipeline (as predicted under steady state conditions) less the amount of liquid that flows out of the line during the transit time for the pig.

A pig is likely to average between 50% and 80-90% of the steady state gas velocity in the pipeline.

Consequently, the success of the calculation depends on obtaining a suitable estimate for the pig transit time. As an upper bound, the pig cannot travel faster than the average steady state gas velocity. In fact, it is reasonable to assume that even in a relatively flat pipeline, it would be unlikely to average *more* than 80% to 90% of the average steady state gas velocity. There will be a pressure loss across the pig itself and an increasing resistance to flow as liquid accumulates ahead of it. On the other hand, it is relatively unlikely to average *less* than 50% of the steady state gas velocity, as the pressure buildup behind the pig would become quite substantial. In hilly terrain, with moderate to large liquid-to-gas ratios, the pig may well travel at somewhere around 60% of the average steady state gas velocity.

Unless the pipeline is quite long with relatively high liquid loading, the differences in slug size due to pig velocity are not excessive. For most cases, this check can provide a useful guide for sizing the liquid receiving facilities.

A Pig Launcher allows you to begin a Pigging Slug Check in the middle of a pipeline.

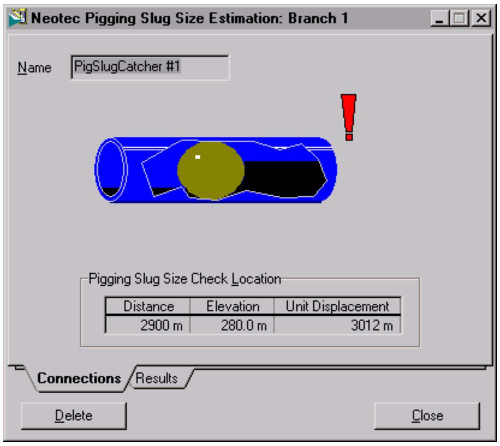
If a Pig Launcher is not added to the elevation profile, the test is taken from the beginning of the profile to the pig slug catcher.

The insertion of a **Pigging Slug Check** in a pipeline profile indicates the termination point of the test, or the location of the pig slug catcher. Therefore, the length of the pipeline through which the pig travels

spans the distance between its insertion point at a **Pig Launcher** and the **Pigging Slug Check**. The **Pig Launcher** is available on the drop-down list of in-line facilities options on the Elevation Profile tab. It is added to the Elevation Profile tab in the same manner as any other in-line facility. Because the **Pig Launcher** serves only as a marker for the beginning of a Pigging Slug Check, it does not have any physical properties and therefore does not have an associated view.

You are able to add more than one Pig Launcher to the elevation profile of a PIPESYS Extension. However, PIPESYS will perform the slugging check between the pig slug catcher and the nearest upstream **Pig Launcher**. If you do not insert a **Pig Launcher** into the profile, PIPESYS will calculate the volumes from the beginning of the pipeline. Use the ***Cut*** button on the Elevation Profile tab to remove any undesired Pig Launchers.

Figure 9.6



9.6.1 Connections Tab

As on all component views, the location for the unit is displayed as read-only data. If you need to change this data, open the **Main PIPESYS View** and go to the **Elevation Profile** tab.

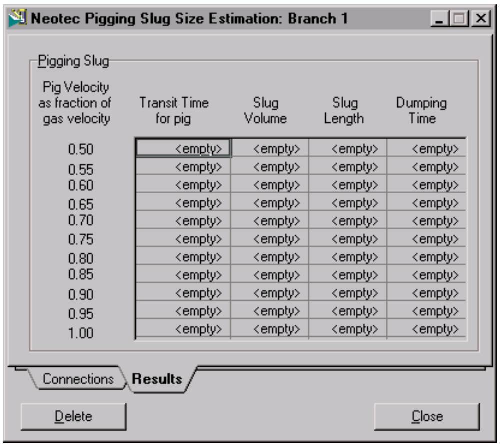
9.6.2 Results Tab

There are no data entry requirements for this calculation. Parameters are all taken from the calculated conditions in the pipeline.

The **Results** tab provides three measures of pigging slug size for a range of pig velocities. You must use your own judgement to decide which pig velocity is likely to match the actual situation.

For each value of pig velocity, the pig transit time, slug volume, slug length and slug dumping time are displayed in the results matrix. The transit time is the time taken for the pig to travel from the launching point to the check location. Slug volume is the volume of liquid ahead of the pig as discussed above and slug length measures how much of the pipeline ahead of the pig is occupied by the slug, assuming that the pipeline is completely filled with liquid. Dumping time is the time required for the slug to flow out of the pipeline under steady state conditions.

Figure 9.7



9.7 Severe Slugging Check

Severe slugging is associated with veritcal or near-vertical risers, which are a common feature of offshore production platforms.

The **Severe Slugging Check** is an application of two different criteria to predict the likelihood of severe slugging at a particular point in the pipeline.

Severe slugging is a phenomenon associated with vertical or near-vertical risers, which are a common feature of pipelines connected to offshore production platforms but may also be encountered in pipelines that traverse uneven terrain. Severe slugging occurs only at points where a steep riser, inclined at 70^{o}or more, is immediately

downstream of a gradually descending section of the pipeline and even then, only under conditions of low to medium gas and liquid flow rates.

If the flow is stratified, a liquid seal may form at the base of the riser and block the gas flow. The liquid will continue to flow and accumulate in the riser, forming a slug. This slug will grow and expand to fill the riser if the rate at which the hydrostatic head of the slug increases is faster than the rate at which the gas pressure increases upstream of the slug.

Eventually the liquid will reach the top of the riser and continue to flow through the pipeline.   However, this situation does not represent an equilibrium state because the gas flow remains blocked. The hydrostatic head of the slug cannot further increase but the pressure due to the buildup of gas at the base of the riser will finally exceed the hydrostatic head of the slug and cause gas to enter the riser. Sometimes the gas pressure can exceed the hydrostatic head of the slug before the liquid reaches the top of the riser, but the end result is the same and gas moves into the riser.

As the gas displaces the liquid in the riser, the hydrostatic head is reduced, causing a corresponding expansion of the gas. At some point the remaining liquid in the riser will be energetically blown downstream and the gas pressure will drop to its minimum value. Then the liquid-buildup begins anew and the process repeats.

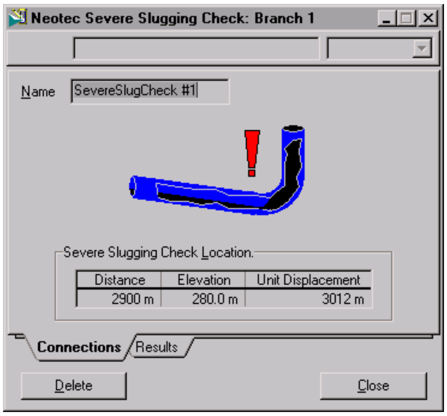
Severe slugging can be a problem for pipeline engineers and operators because of the irregular flow pattern that develops. The resultant variations in pressure and flow rate make for an ill-behaved system that is difficult to control and operate according to requirements.

If possible, severe slugging should be prevented. It is less likely to occur at higher pressures and can be inhibited by choking the flow at the top of the riser, although at the cost of reducing production rates. Other prevention methods include installing an active flow control device at the top of the riser, or injecting gas into the riser just above the base. This ensures that the riser can never completely fill with liquid.

9.7.1 Connections Tab

As on all component views, the location for the unit is displayed as read-only data. If you need to change this data, open the **Main PIPESYS View** and go to the **Elevation Profile** tab.

Figure 9.8



9.7.2 Results Tab

There are no data entry requirements for this calculation. Parameters are taken from the calculated conditions in the pipeline.

PIPESYS computes two different severe slugging criteria, based on conditions in the system at the point of interest. If the actual value for the criterion is less than the critical value, the model predicts that severe slugging occurs, by filling in the check box in the **Slugging?** column. Additionally, it should be noted that in the absence of a truly definitive criterion, severe slugging should be predicted to be a potential problem unless both Pots (1985) and Fuchs (1987) models say otherwise.

If the current pipeline profile does not incorporate the geometry necessary for severe slugging to occur, a warning will be issued by the software and appear on this tab.

9.8 Erosion Velocity Check

Erosion damage (i.e. the wearing away of material) may occur as a result of the impact of high velocity liquid droplets. It may also be caused by solid particles, such as sand, entrained in the gas or liquid stream. Erosion damage is typically controlled by limiting the maximum gas or liquid velocity in the system. PIPESYS can calculate such limiting values for a wide range of flowing conditions.

Erosion losses can be significant in sections of pipe where the flow abruptly changes direction.

Erosion caused by liquid droplets is primarily a concern in gas-condensate or gas-water systems, but may also be a concern in oil-gas systems where there is a very high gas-oil ratio, as occurs with wells under gas lift. Erosion losses are seldom a problem in a straight run of pipe but can be significant anywhere the flow abruptly changes direction (e.g. manifolds, elbows, tees, etc.). It is usual to perform this check at the downstream end of the pipeline where, not only do such direction changes typically occur, but generally also the highest gas velocities prevail.

For liquid-droplet erosion caused by a sand-free fluid, the limiting velocity is defined by the following empirical equation,

$$V_M = \frac{C}{\sqrt{\rho_M}}$$

where: C *= a constant*

ρ_M = *mixture density, lb/ft*^{3}

V_M = *maximum allowable mixture velocity*

The PIPESYS erosion velocity check makes this calculation for two values of the constant C so that you may choose a more conservative or less conservative maximum velocity depending on your need to limit erosion. To eliminate erosion losses it is suggested that the value recommended by the API RP14E (C = 100) be used. For situations in which a small amount (less than 10 mils per year) of erosion can be tolerated, Salama and Venkatesh recommend a higher value (C = 300). It is important to realize that the minimum erosion velocity is a function of the gas density. Much higher velocities can be tolerated at low pressure than at high pressure.

A different relation is used for fluids bearing sand. The maximum allowable velocity is obtained using,

$$V_M = \frac{4d}{\sqrt{W_s}}$$

where: d = pipe inside diameter (inches)

W_s = rate of sand production (bbl/month) where 1 bbl sand = 945 lb (429 kg)

V_M = maximum allowable mixture velocity

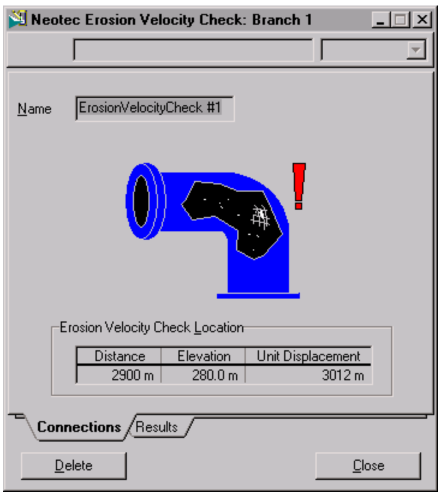
The maximum velocity as Ws approaches zero is assumed to be constrained by the previous equation.

Even relatively small amounts of sand production have a strong influence on the maximum velocity that should be permitted. For example, with 4" I.D. tubing and a typical gas density of 4.0 lb/ft^3, the effect of sand production dominates the allowable velocity at any sand production rate above about 100 lb/month.

9.8.1 Connections Tab

As on all component views, the location for the unit is displayed as read-only data. If you need to change this data, open the **Main PIPESYS View** and go to the **Elevation Profile** tab.

Figure 9.9



9.8.2 Results Tab

The **Fluid Conditions** group box displays the Actual Gas Velocity, Mixture Velocity, Mixture Density and Effective C Value at the erosion velocity check location. The mixture velocity data is provided so that you may compare this value against the allowable velocities that appear in the matrix in the **Allowable Velocity** group box. The actual gas velocity and mixture density values are provided for your information.

To analyse the data provided in this tab, you need some information about your system. You will need an estimate of sand production for the system in units of lb/month or kg/month and you need to evaluate the sensitivity of the system to erosion damage. If you wish to eliminate erosion, choose C = 100. If this condition is too restrictive in terms of allowable velocity and a small erosion rate over the life of the pipeline can be tolerated, use C = 300.

Using the actual mixture velocity as an assumed erosion velocity, an effective C value is computed assuming no sand production. This gives a measure of a minimum value for C for which erosion will not be a concern for this case.

Find the value of sand production in the Sand Production column that is closest to the value for your system. This is expressed in lb/month or kg/month. Read across to the allowable velocity value that corresponds to the value of C that you have chosen. If this allowable velocity value is less than the Mixture Velocity in the Fluid Conditions group box, erosion is likely to be higher than acceptable and you should make whatever changes are necessary to lower the mixture velocity at this location. If the allowable velocity value is greater than the mixture velocity, erosion is unlikely to be a problem.

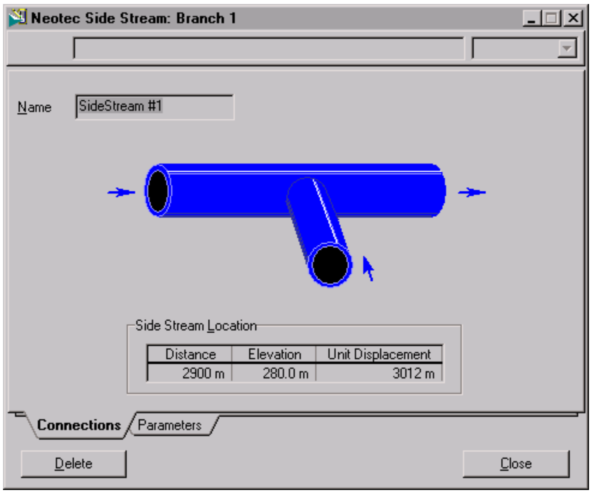
9.9 Side Stream

A Side Stream Pipeline Unit in PIPESYS can be used to add or remove flow from the elevation profile. For pipelines with multiple sources, this provision eliminates the need to create multiple streams and multiple pipeline units. This is especially useful if pressure losses in the side stream are either inconsequential (very short lines) or irrelevant to the analysis. What would otherwise be a complex system of pipes, can often be reduced to a few pipelines with side streams. Data for the calculations are entered in the Parameters tab of the Side Stream Pipeline Unit and main line results are displayed for the streams before and after the side stream.

9.9.1 Connection Tab

As on all component views, the location for the unit is displayed as read-only data. If you need to change this data, open the **Main PIPESYS View** and go to the **Elevation Profile** tab.

Figure 9.10



9.9.2 Parameters Tab

This page is used to define the basic characteristics of the side stream. In the Flow Direction group box there are two radio buttons from which you can choose the direction of the flow: Inflow or Outflow. Selecting the flow direction and flow rate specifies addition or removal of flow. The side stream's flow rate can be specified as a:

- molar flow rate
- mass flow rate
- or a standard liquid volume flow rate.

For an inflow stream when calculating the flowing fluid temperature profile, the temperature or vapour mole fraction of the stream can be optionally specified. If the flowing fluid temperature profile is being specified or the temperature or vapour mole fraction are left empty, then the inflow stream's temperature and vapour mole fraction are obtained from main line stream conditions. The Before Side Stream main line stream's data is used when calculations are done in the

direction of flow. For calculations done against the direction of flow the After Side Stream main line stream’s data is used.

The outflow stream always obtains its temperature and vapour mole fraction data from the main line streams. Similarly, as when the inflow stream’s temperature and vapour mole fraction data are left empty the Before Side Stream main line stream’s data is used when calculations are done in the direction of flow. For calculations done against the direction of flow the After Side Stream main line stream’s data is used.

10 Gas-Condensate Tutorial

10.1 Setting Up the Flowsheet 3

10.2 Adding a PIPESYS Extension 8

10.3 Applying a Global Change.. 16

This example guides you through the construction of a gas-condensate pipeline consisting of four Pipe Units. A Fluid System with a hypothetical component is used in a pressure drop calculation for a predetermined flow rate through the pipeline. All units for this example are Field.

10.1 Setting Up the Flowsheet

The following table shows the fluid package you will create for this example:

Property Package	Components
Peng Robinson	C1, C2, C3, i-C4, n-C4, i-C5, C6, C7+*, Nitrogen, CO2, H2S

To change the unit set to Field, go to Tools in the menu bar and choose Preferences. Click on the Units tab to change the unit set.

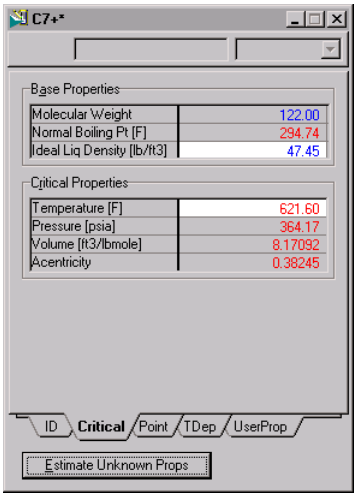
1. Start HYSYS and create a **New** case.
2. Create a Select **EOS**s in the **Property Package Filter** group box and then **PR** in the **Base Property Package Selection** group box.
3. Open the Components tab on the Fluid Package view and use the ***Pure*** button to add **C1, C2, C3, i-C4, n-C4, i-C5, n-C5, C6, Nitrogen, CO2** and **H2S** to the Current Component List.
4. Select the ***Hypothetical*** radio button on the Add Comps group box on the Components tab and then click the ***Quick Create a Hypo Comp...*** button. This will bring up the **Hypothetical Component Property View.**
5. On the **Hypothetical Component Property View** enter **C7+** into the **Component Name** cell and make sure that **Family/Class** is set to **Hydrocarbon**.
6. Open the **Critical** tab of the Hypothetical Component Property View and enter **122** into the **Molecular Weight** cell.
7. Enter **760 kg/m^3** into the **Ideal Liquid Density** cell by first selecting that cell and then typing “760" into the Edit Bar. Select units of kg/m^3 from the Edit Bar drop down list and the program will automatically convert the liquid density to 47.45 lb/ft^3.

The workbook can also be accessed by using the hotkey combination ***<Ctrl><D>***.

For more information on creating a Hypothetical Component refer to ***Chapter 9 - Hypotheticals*** *of the HYSYS Reference, Volume 1.*

8. Finally, press the ***Estimate Unknown Props*** button to complete specification of the hypothetical component. Verify that the **Critical** tab appears as in Figure 10.1 before closing the view.

Figure 10.1

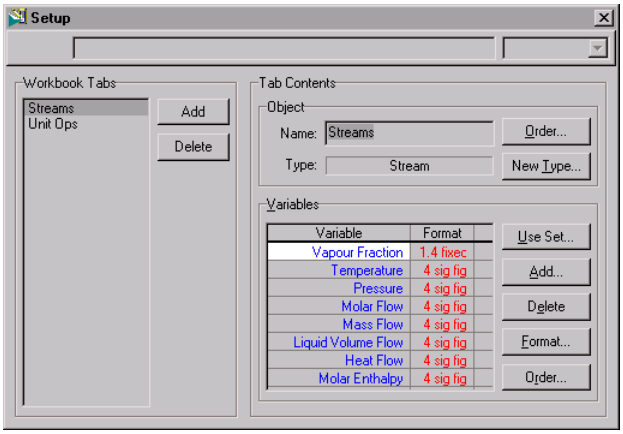


9. On the **Fluid Package** view, select the C7+ component in the **Hypo Components** list and press the ***Add Hypo*** button to add it to the **Current Component List** to complete the specification of the fluid.
10. **Close** the Fluid Package view.
11. Press the ***Enter Simulation Environment...*** button at the bottom of the **Simulation Basis Manager** view.
12. Open the **Workbook**.

Workbook button

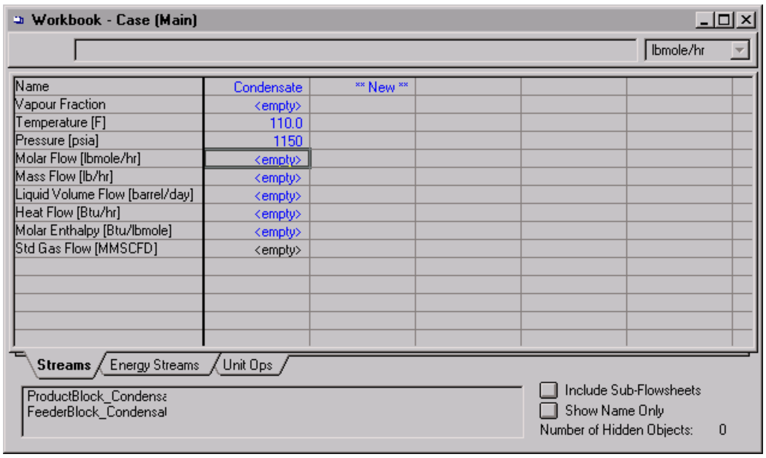
13. To change the Workbook display, select **Workbook** from the Main Menu bar and then **Setup** to display the **Setup** view (Figure10.2). Press the ***Add...*** button in the Variables group box to display the Select Variable(s) for Main dialog box. Select **Std Gas Flow** in the Variable(s) list. Press the ***OK*** button in the **Select Variable(s) for Main** dialogue box. **Close** the **Setup** view.

Figure 10.2



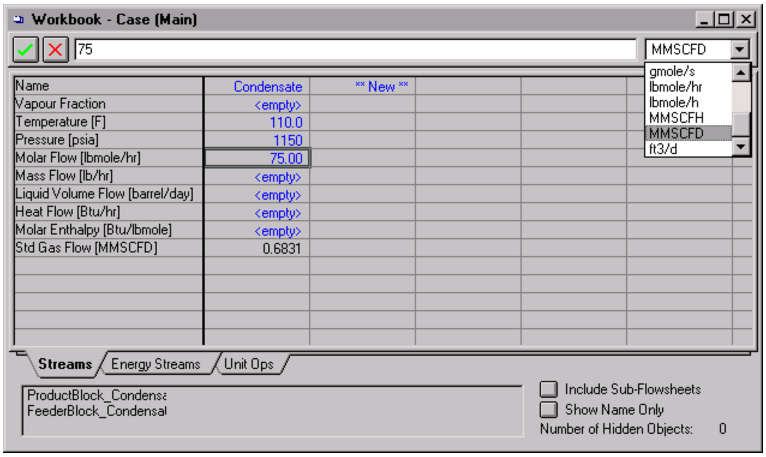
14. Create a new **Material Stream**. Name it **Condensate** and type **110 F** into the **Temperature** cell and **1150 psia** into the **Pressure** cell. See Figure 10.3 below:

Figure 10.3



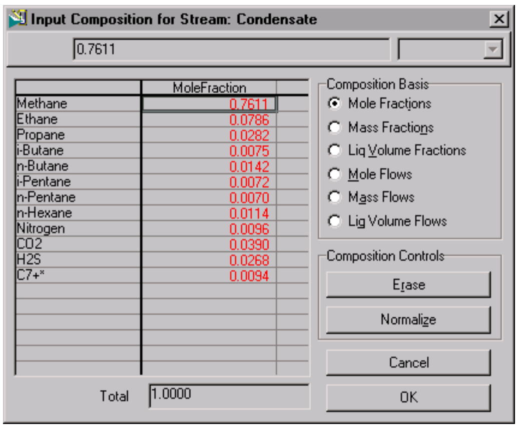
15. Enter **75 million standard cubic feet per day** (**MMSCFD**) into the **Molar Flow** cell on the **Workbook** view (Figure 10.4). HYSYS will convert this value to 8235. lbmole/hr.

Figure 10.4



16. Double click the **Molar Flow** cell and the dialog box **Input Composition for Stream: Condensate** will open. Complete the composition detailed on the following tab as shown in Figure 10.5 and click ***OK***.

Figure 10.5



17. Create a second **Material Stream**, which will be the outlet stream of the pipeline, by typing. Name this stream **Outlet**.
18. Open the Energy Streams tab of the Workbook view. Define an **Energy Stream** by entering the name **Pipeline Energy Transfer** into the **Name** cell.

The following table summarizes the required input for the **Condensate** stream:

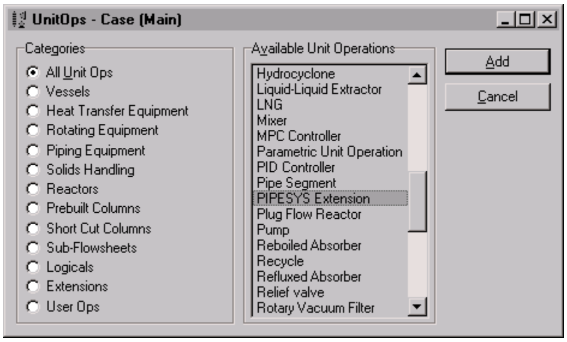
*** signifies required input*

Name	Condensate
Vapour Fraction	0.9576
Temperature [^{o}F]	110**
Pressure [psia]	1150**
Molar Flow [lbmole/hr]	8235**
Mass Flow [lb/hr]	1.905e+05
Liq Volume Flow [barrel/day]	3.399e+04
Heat Flow [Btu/hr]	-3.307e+08
Std Gas Flow [MMSCFD]	75.00**
Comp Mass Frac [Methane]	0.76110**
Comp Mass Frac [Ethane]	0.07860**
Comp Mass Frac [Propane]	0.02820**
Comp Mass Frac [i-Butane]	0.0075**
Comp Mass Frac [n-Butane]	0.0142**
Comp Mass Frac [i-Pentane]	0.0072**
Comp Mass Frac [n-Pentane]	0.0070**
Comp Mass Fac [n-Hexane]	0.0114**
Comp Mass Frac [Nitrogen]	0.0096**
Comp Mass Frac [CO2]	0.0390**
Comp Mass Frac [H2S]	0.0268**
Comp Mass Frac [C7+*]	0.0094**

10.2 Adding a PIPESYS Extension

1. Add the PIPESYS Operation to the HYSYS case by selecting **Flowsheet** and then **Add Operation...** from the Menu Bar.
2. Select **PIPESYS Extension** from the **Available Unit Operations** list in the **UnitOps** dialog, as shown in Figure 10.6, and press the ***Add*** button.

Figure 10.6

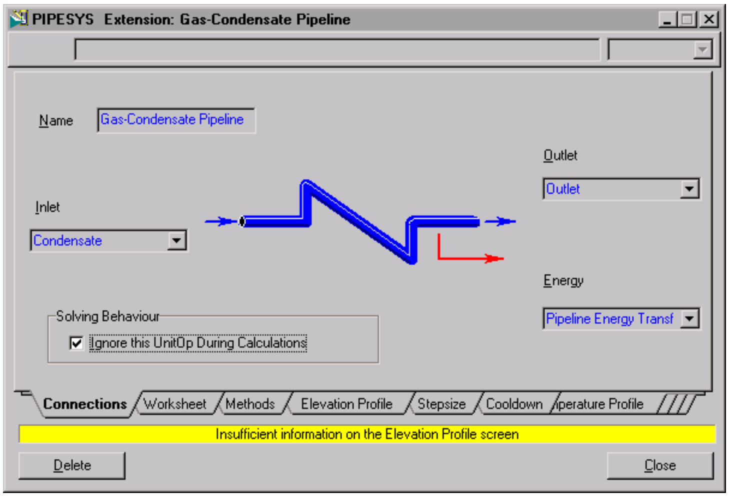


The **Main PIPESYS View** should now be on your screen, displaying the **Connections** tab.

3. Name the PIPESYS extension **Gas-Condensate Pipeline**.
4. From the **Inlet** drop down list, select the **Condensate** stream.
5. Select the **Outlet** stream from the **Outlet** drop down list.
6. Select **Pipeline Energy Transfer** from the **Energy** drop down list.
7. Click on the **Ignore this UnitOp During Calculations** check box.

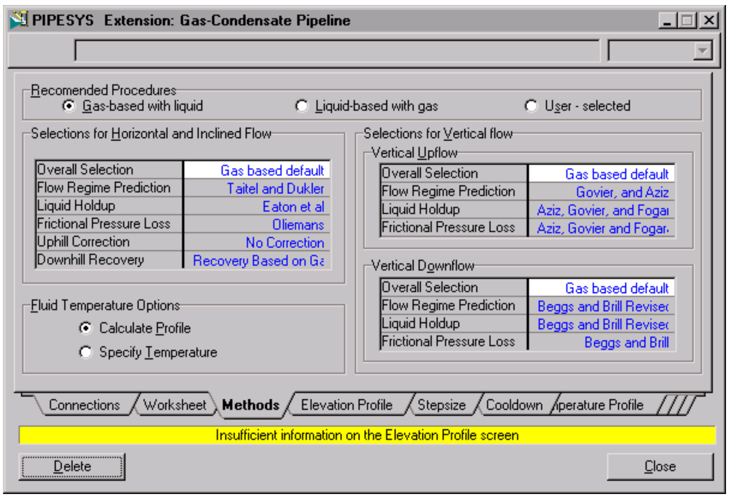
The **Ignore the UnitOp During Calculations** allows you to disable the concurrent calculation of intermediate results while you are specifying data to the PIPESYS extension. Figure 10.7 shows the completed view.

Figure 10.7



8. Open the **Methods** tab. Make sure the ***Gas-based with Liquid*** and the ***Calculate Profile*** radio buttons are selected as in Figure 10.8.

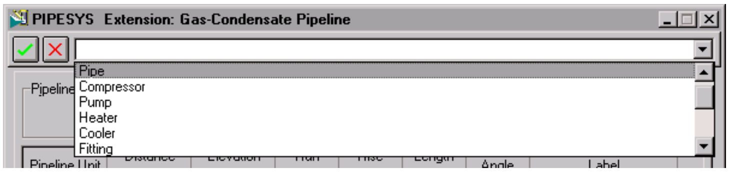
Figure 10.8



9. Select the **Elevation Profile** tab. Here you will define the geometry and physical characteristics of the pipeline. Enter **0** into the **Distance** cell and **2800** into the **Elevation** cell in the **Pipeline Origin** group box.
10. Select the cell in the Pipeline Unit column that reads **<empty>**. Then from Edit Bar at the top of the view, select **Pipe** from the drop down list, as in Figure 10.9.

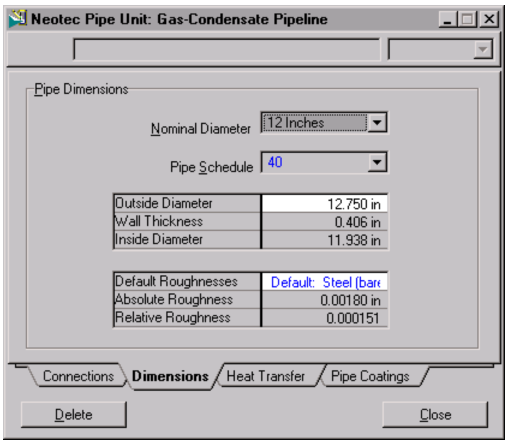
A **Pipe Unit** will be added to the elevation profile matrix and the **Pipe Property View** will open.

Figure 10.9



11. Open the **Dimensions** tab of the **Pipe Property View**. Select **12 Inches** from the **Nominal Diameter** drop down list. Select **40** from the **Pipe Schedule** drop down list. When you are finished these steps, the Dimensions tab will appear as in Figure10.10.

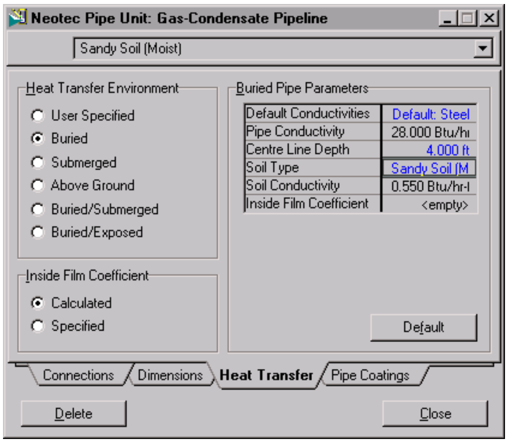
Figure 10.10



12. Open the **Heat Transfer** tab of the Pipe Property View. Enter **4** ft for the **Centre Line Depth** parameter.

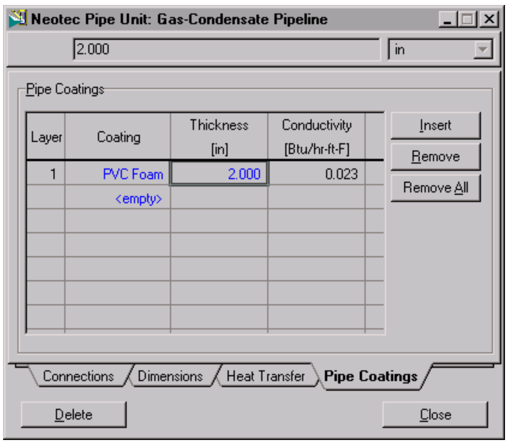
13. Choose **Sandy Soil (Moist)** from the drop down list for **Soil Type.**

Figure 10.11



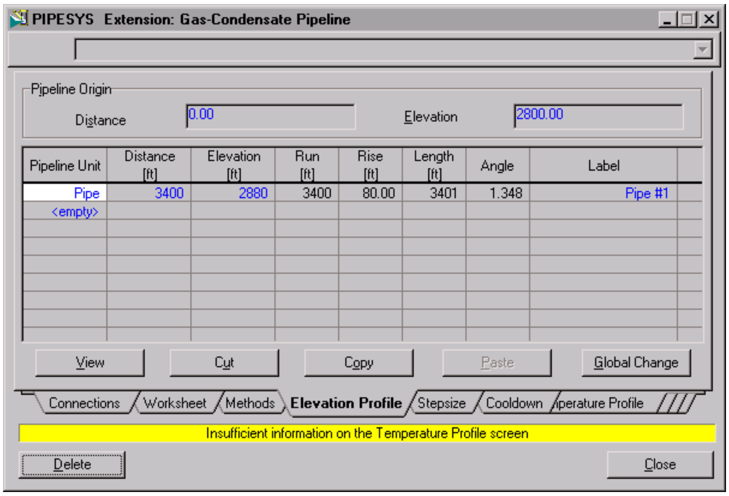
14. Go to the **Pipe Coatings** tab of the **Pipe Property View**. Add a single layer of insulation consisting of **PVC Foam** with a thickness of **2 inches**. See Figure 10.12 to verify the correctness of your data entries before pressing the ***Close*** button on the **Pipe Property View**.

Figure 10.12



15. Complete the specification for the first **Pipe Unit** by entering **3400 ft** into the **Distance** column and **2880 ft** into the **Elevation** column of the elevation profile matrix. Figure 10.13 shows the data entry completed for the first **Pipe Unit.**

Figure 10.13

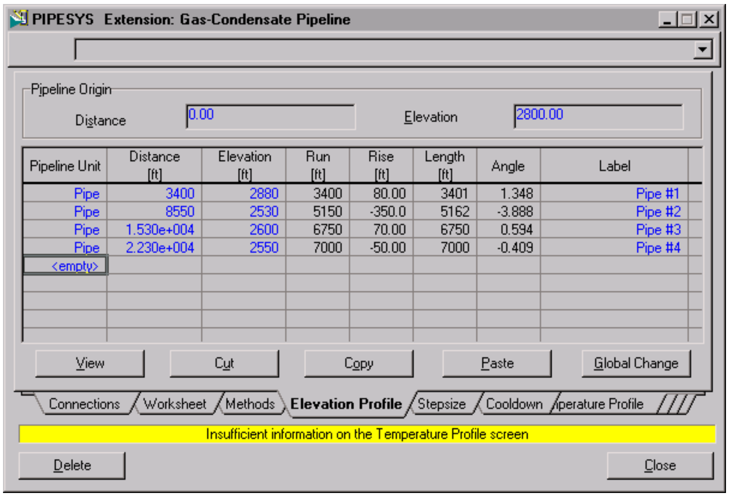


Using the ***Copy*** and ***Paste*** buttons on the **Elevation Profile** you are able to copy existing Pipeline Units from the Elevation Profile tab and create a new Pipeline Unit with identical properties. This saves time when creating a pipeline consisting of several identical pipe units.

16. Select the **Pipe Unit** that you want to copy; in this case it is Pipe #1, and press the ***Copy*** button. You will notice that the ***Paste*** button, previously greyed out, becomes active.
17. Select the cell in the Pipeline Unit column with the **<empty>** label and press the ***Paste*** button. A new **Pipe Unit** will be added to the profile. Repeat this procedure twice so that the elevation profile matrix has a *total of four Pipe Units.*

18. Since the ***Copy*** and ***Paste*** procedure copies only the property view data for the Pipe Units, you are required to enter the elevation profile data for the remaining three units. Use the data values shown in Figure 10.14 to fill in the **Distance** and **Elevation** parameters.

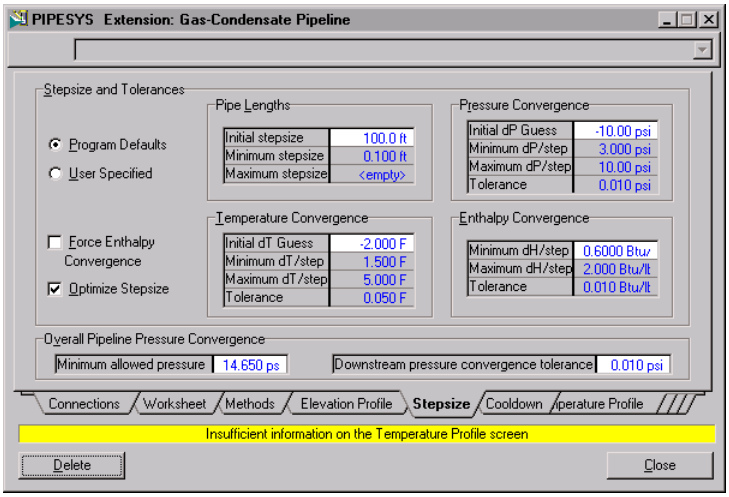
Figure 10.14



For most cases, the PIPESYS default Stepsize and tolerance values are acceptable for the extension calculations.

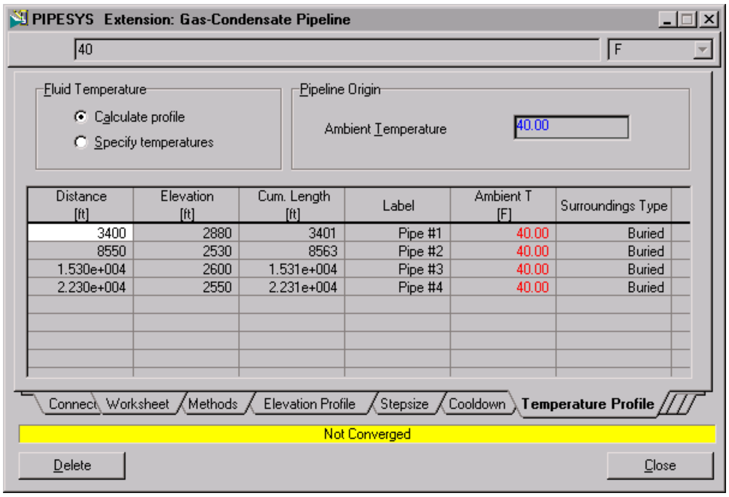
19. Open the **Stepsize** tab of the Main PIPESYS View. Make sure the ***Program Defaults*** radio button is selected, as in Figure 10.15.

Figure 10.15



20. Open the **Temperature Profile** tab of the Main PIPESYS View.
21. Enter **40 F** into the **Ambient Temperature** cell in the **Pipeline Origin** group box as shown in Figure 10.16.

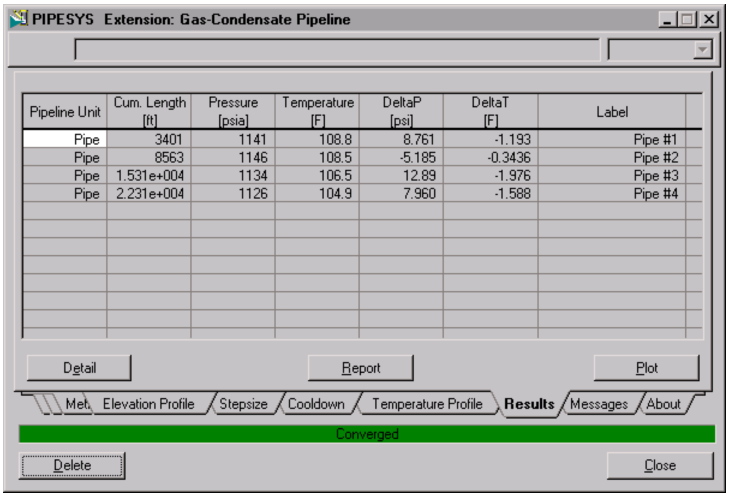
Figure 10.16



22. Return to the **Connections** tab of the Main PIPESYS View. Since data entry is complete, you can instruct the program to begin calculations by removing the check in the **Ignore this UnitOp During Calculations** check box. After a few seconds, the program will find a solution and announce success by displaying “Converged” on the Object Status at the bottom of the Main PIPESYS View.

23. You can view the results for each Pipe Unit on the **Results** tab on the Main PIPESYS View. This tab shows pressure losses, fluid temperatures and liquid holdup for each Pipe Unit. See Figure 10.17.

Figure 10.17



24. You can view more detailed results by pressing the ***Detail*** button on the **Results** tab. This will bring up the **Pipe Segment Results** view which displays a comprehensive array of hydraulic data for each calculation segment.

25. Check to see if the program encountered any difficulties during the calculation phase by opening the **Messages** tab. For this case, there should have been no difficulties and the tab should read, “No calculation warnings encountered.”

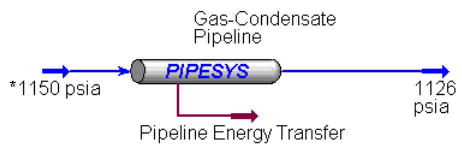
26. To open the **PFD**, press the ***PFD*** button on the Button Bar. Right click anywhere on the PFD view to obtain a pop-up menu.

27. Select **Choose Label Variable** from the menu and the **Select Variable** for PFD Labels dialog box will appear. You can use this dialog to display a number of process variables right on the PFD.

28. Press the ***Change Variable*** button and choose **Pressure** from the Choose Label Variable dialog box.

29. Press the ***OK*** button on the **Choose Label Variable** dialog box and the inlet and outlet pressures will be displayed on the PFD. See Figure 10.18.

Figure 10.18



30. To print the PFD schematic, right click anywhere on the PFD and select the **Print PFD** command from the pop-up menu.

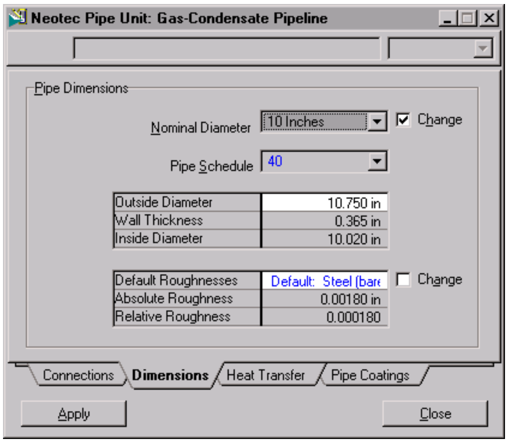
10.3 Applying a Global Change

By using the **Global Change** feature, you can quickly change the pipe size for all pipe units and then let PIPESYS recalculate the flow parameters for the extension.

1. Open the **Elevation Profile** tab on the Main PIPESYS View, select the first pipe in the list and press the ***Global Change*** button.
2. Select the **Dimensions** tab. Choose **10 Inches** from the **Nominal Diameter** drop down list and select **40** from the **Pipe Schedule** drop down list.

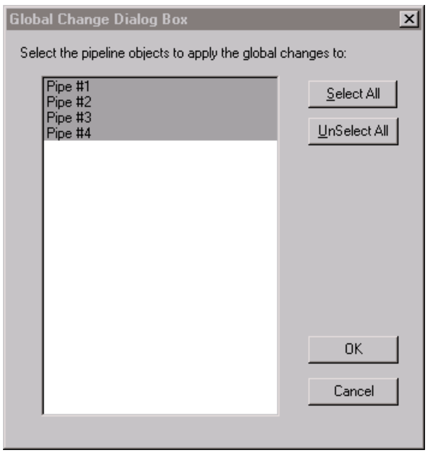
You should notice that the **Change** check box beside the Nominal Diameter drop down list became checked as soon as you made these changes. This is to notify you that the program is aware that this parameter has changed and that this change can be duplicated for other Pipe Units in the pipeline.

Figure 10.19



3. Press the ***Apply*** button on the **Global Change** property view.
4. The **Global Change Dialogue Box** will appear. This dialogue box allows you to specify which Pipe Units will be subject to the changes. In this case, you are changing all of them, so press the ***Select All*** button (see Figure 10.20) and press the ***OK*** button.

Figure 10.20



5. Press the ***Close*** button on the **Pipe Property** view.The changes will be registered with the program but will not be implemented until this view is closed.

6. The program will immediately start to recalculate for the 10" diameter. When the **Object Status** displays **Converged**, you can look at the calculated results and compare them with the values obtained for the 12" pipe.

You have now completed the Gas-Condensate pipeline example. For a more in-depth exercise in using the PIPESYS Extension, see **Gas-Condensate Gathering System** and **Optimizing the Gas-Condensate Gathering System** in the **PIPESYS Applications Binder**.

11 PIPESYS Application 1

11.1 Gas Condensate Gathering System 3

11.2 Setting up the Flowsheet 6

11.3 Setting Up the Case........ 8

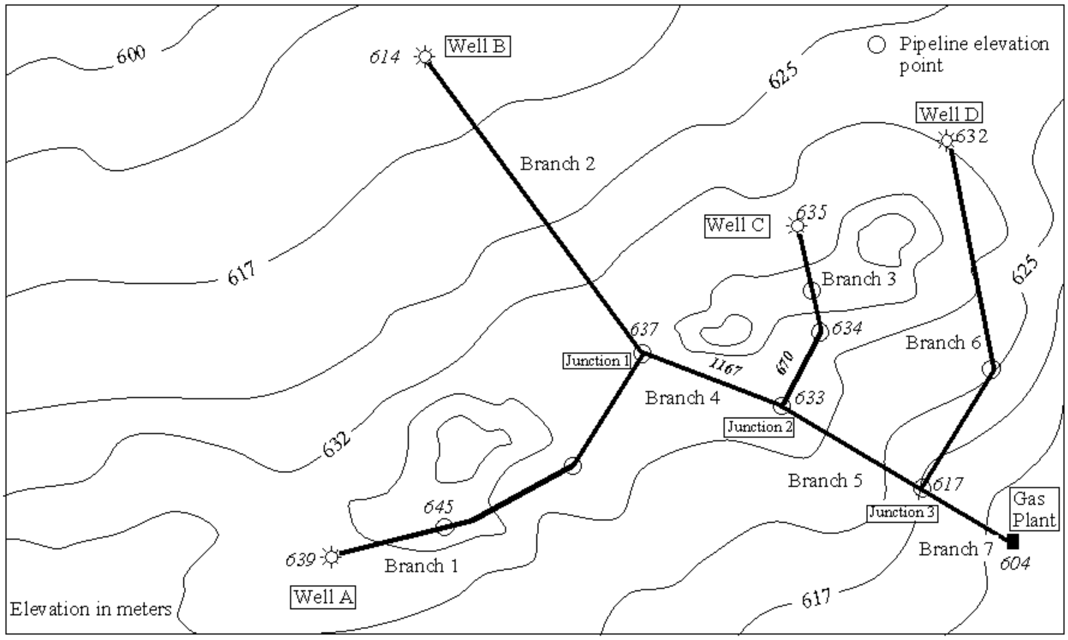
11.3.1 Adding the First PIPESYS Extension 8
11.3.2 The Second PIPESYS Extension........ 12
11.3.3 Adding a Mixer 13
11.3.4 The Third PIPESYS Extension........ 14
11.3.5 The Fourth PIPESYS Extension........ 15
11.3.6 The Fifth PIPESYS Extension 16
11.3.7 The Second Mixer 17
11.3.8 Well Stream Information........ 18

11.4 Results........ 18

11.1 Gas Condensate Gathering System

In this PIPESYS Application, the performance of a small gas-condensate gathering system is modelled. Figure 11.1 shows the physical configuration of this system superimposed on a topographic map. The system consists of three wells distributed over an area of approximately 1.0 square mile connected to a gas plant via a network of pipelines.

Figure 11.1



Field data shows that the wells are delivering the following rates:

Well A	8.6 MMSCFD
Well B	7.4 MMSCFD
Well C	10.1 MMSCFD

All three wells have the same composition. A residual of all the heavier components in the condensate has a molecular weight of 122 and a density of 760 kg/m^3. The characteristics of this component will be accounted for by using the hypothetical component facility in HYSYS.

The compositional analysis of the gas-condensate resulted in the following information:

Methane	0.623	**n-Pentane**	0.00405
Ethane	0.280	**n-Hexane**	0.00659
Propane	0.0163	**C7+**	0.00992
i-Butane	0.00433	**Nitrogen**	0.00554
n-Butane	0.00821	**Carbon Dioxide**	0.0225
i-Pentane	0.00416	**Hydrogen Sulfide**	0.0154

Pipe diameters for each of the branches are:

Branch 1	3”
Branch 2	3”
Branch 3	3”
Branch 4	4”
Branch 5	6”

Schedule 40 steel pipe is used throughout and all branches are buried at a depth of 3 feet. All pipes are uninsulated.

Elevation data for each of the branches are provided in the following table. The elevation given for the pipe units is for the *endpoint* of the pipe, i.e. the downstream end. Branches that traverse undulating terrain have been subdivided into a number of segments with elevation points assigned at locations where there is a significant slope change. Such locations in the network are labelled on the schematic diagram with the elevation value in italics. The following table summarizes the elevation data. For each of the branches, the resulting distance and elevation data as obtained from the topographic map is listed. With these data, you can simulate the performance of the given system using PIPESYS extension and thereby calculate important parameters such as pressure losses, temperature changes and liquid-holdup amounts as well as predicting the flow regimes.

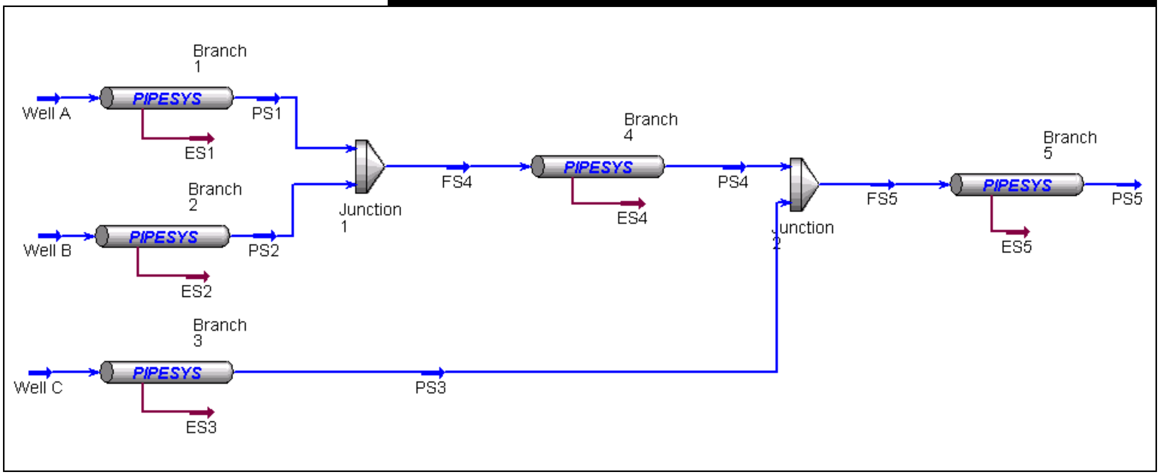
Branch	Unit	Length (ft)	Elevation(ft)
Branch 1	Well A	n/a	2095
	Pipe Unit 1	945	2110
	Pipe Unit 2	1110	2089
	Pipe Unit 3	1056	2090

Branch	Unit	Length (ft)	Elevation(ft)
Branch 2	Well B	n/a	2015
	Pipe Unit 1	2822	2090
Branch 3	Well C	n/a	2085
	Pipe Unit 1	528	2125
	Pipe Unit 2	334	2080
	Pipe Unit 3	670	2077
Branch 4	Pipe Unit 1	1167	2077
Branch 5	Pipe Unit 1	2110	1980

In this simple example, the flow rate at each well is specified and is independent of the flow rate at each of the other wells. In cases such as this, the system can be modelled with only one pressure drop determination per branch. Simultaneous pressure and temperature calculations can be performed if the temperature at each wellhead is also known.

Figure 11.2 shows the PFD generated by HYSYS for the completed case.

Figure 11.2



Since pressures are continuous throughout the network, the pressure can be specified at only one point. For instance, the pressure can be fixed at any one well or at the final delivery point and PIPESYS will compute the pressure everywhere else. For this application example, a pressure of 1060 psia will be specified for Well A. PIPESYS will then determine the pressures elsewhere in the network that are consistent with this specification.

Heat transfer calculations should be performed in the direction of flow whenever possible. Furthermore, wellhead temperatures are generally known. For this example, the fluid temperatures at wells A, B and C are known and must be entered as fixed conditions. PIPESYS will then perform an iterative pipeline calculation in branches where the upstream temperature and downstream pressure are known. Temperatures of the blended fluids will be computed on a mass basis downstream of the junctions of two or more streams.

11.2 Setting up the Flowsheet

Carry out the following steps to model the gathering system with PIPESYS:

*For additional information on HYSYS views and conventions, see the HYSYS Reference Manual 1, **Chapter 1 - Interface.***

1. Start HYSYS and create a **New** case. In the Simulation Basis Manager, create a fluid package using the **Peng Robinson** equation of state and consisting of the pure components: **methane, ethane, propane, i-butane, n-butane, i-pentane, n-pentane, hexane, nitrogen, carbon dioxide and hydrogen sulfide**.

Property Package	Pure Components
Peng Robinson	C1, C2, C3, i-C4, n-C4, i-C5, n-C5, C6, Nitrogen, CO2, H2S

2. Create a hypothetical component, **C7+**, with the following user-defined properties. Add it to the fluid package before entering the **Main Simulation Environment**.

*** signifies required input*

Name	C7+**
Molecular Weight	122**
Ideal Liquid Density [lb/ft3]	47.45**

3. Open the Workbook and add the 10 **Material Streams** listed below:

Material Streams	Well A, Well B, Well C PS1, PS2, PS3, PS4, PS5 FS4, FS5

4. Enter the compositional data for **Well A** as specified in the following table:

Methane [mole frac.]	0.623**
Ethane [mole frac.]	0.280**
Propane [mole frac.]	0.0163**
i-Butane [mole frac.]	0.00433**
n-Butane [mole frac.]	0.00821**
i-Pentane [mole frac.]	0.00416**
n-Pentane [mole frac.]	0.00405**
n-Hexane [mole frac.]	0.00659**
C7+ [mole frac.]	0.00992**
Nitrogen [mole frac.]	0.00554**
Carbon Dioxide [mole frac.]	0.0225**
Hydrogen Sulfide [mole frac.]	0.0154**

Because the stream composition is identical over the entire system, the composition of stream specified for Well A may be copied to streams **Well** B and **Well C**.

5. Use the ***Define from Other Stream...*** button on Well B's stream view to define its composition by copying the stream specs from Well A. Repeat for Well C.
6. Now open the **Energy Streams** page on the Workbook view. Enter the names of five energy streams:

Energy Streams	ES1, ES2, ES3, ES4, ES5

For this case, each of the five branches of the pipeline will be represented by a separate PIPESYS extension.

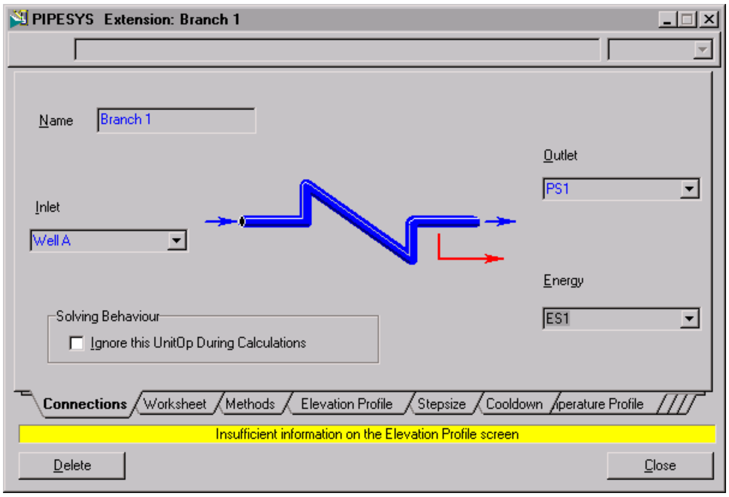
11.3 Setting Up the Case

11.3.1 Adding the First PIPESYS Extension

1. Add a **PIPESYS Extension** to the flowsheet.
2. Complete the **Connections** page as shown in Figure 11.3.

*See the PIPESYS Reference Manual, **Chapter 3 - The PIPESYS View** for a description of all pages in the main PIPESYS view.*

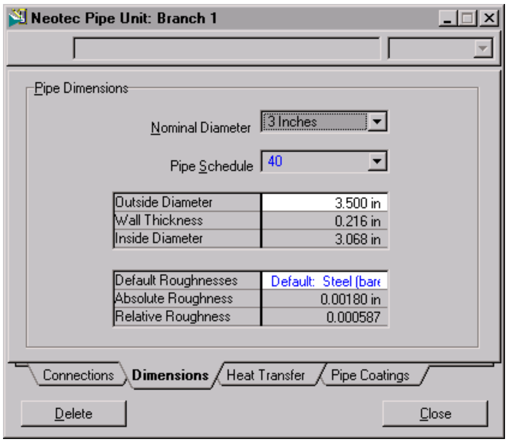
Figure 11.3



3. On the **Elevation Profile** page, and enter **0 ft** into the **Distance** cell and **2095 ft** into the **Elevation** cell.
4. Add the first of three pipe units for this extension on the **Elevation Profile** Page. The **Pipe Unit** view will appear.

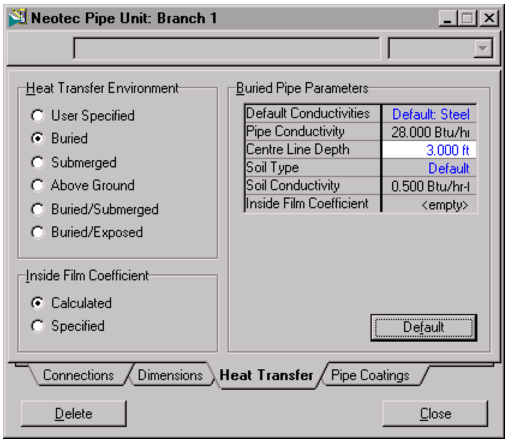
5. On the **Dimensions Page** of the Pipe Unit view, specify the pipe as being **3 inches in diameter**, **schedule 40**. The completed page is shown in Figure 11.4.

Figure 11.4



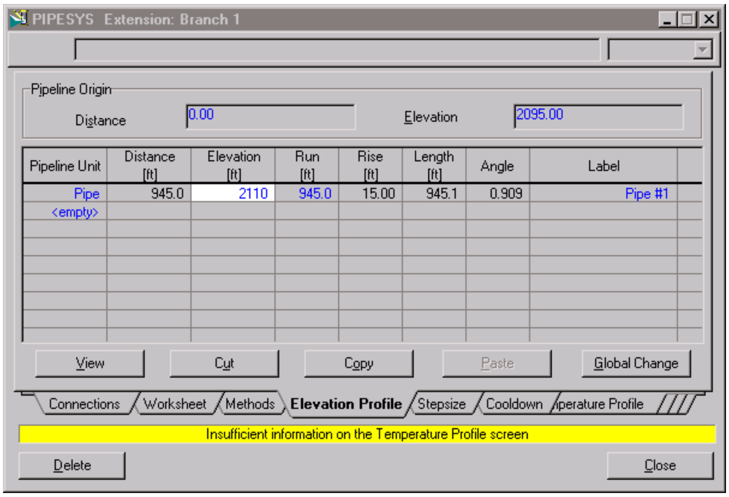
6. On the **Heat Transfer** page, click on the **Centre Line Depth** cell and press the ***Default*** button. All other parameters may be left at their default values. See Figure 11.5 for the completed form.

Figure 11.5



7. **Close** the **Pipe Unit** view and complete the Elevation Profile page by entering **945 ft** for the **Run** parameter and **2110 ft** for the **Elevation** parameter. All other parameters are automatically calculated, as shown in Figure 11.6:

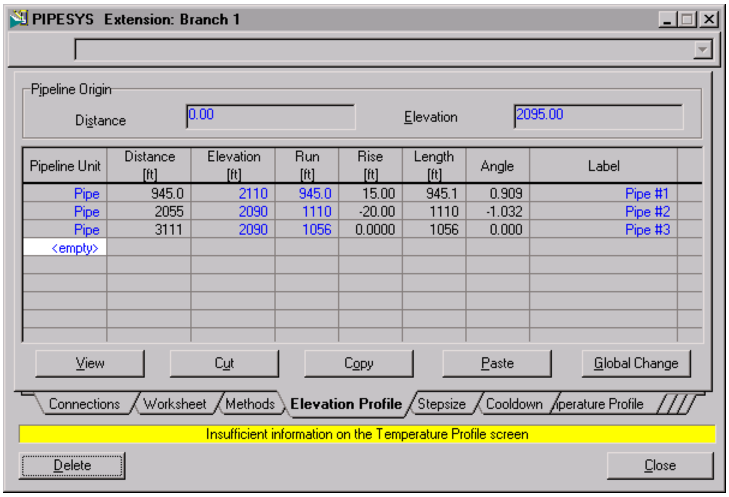
Figure 11.6



8. Add the remaining 2 pipe units. Because all the pipe units for the extension have identical properties to Pipe #1, you may use the ***Copy*** and ***Paste*** buttons as a time saving measure for adding the new units.

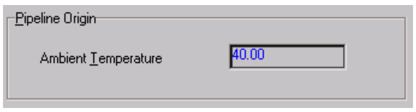
9. Complete the elevation profile as shown by adding the **Elevation** and **Run** parameters for all units. (Figure 11.7)

Figure 11.7



10. Go to the **Temperature Profile** page and enter **40 F** in the **Ambient Temperature** cell in the **Pipeline Origin** group box.

Figure 11.8



This completes the first PIPESYS extension for your case.

11.3.2 The Second PIPESYS Extension

The second PIPESYS extension consists of a single Pipe Unit.

11. Enter the required information for the second extension as defined in the following table:

Connections Page	Name	Branch 2
	Inlet Stream	Well B
	Outlet Stream	PS2
	Energy Stream	ES2
Elevation Profile Page	Distance [ft]	0
	Elevation [ft]	2015
	Pipeline Unit	Pipe #1
	Pipe #1 Elevation [ft]	2090
	Pipe #1 Run [ft]	2822
Pipe Unit View	Nominal Diameter [Inches]	3
	Pipe Schedule	40
	Centre Line Depth	Default
Temperature Profile	Ambient Temperature [F]	40

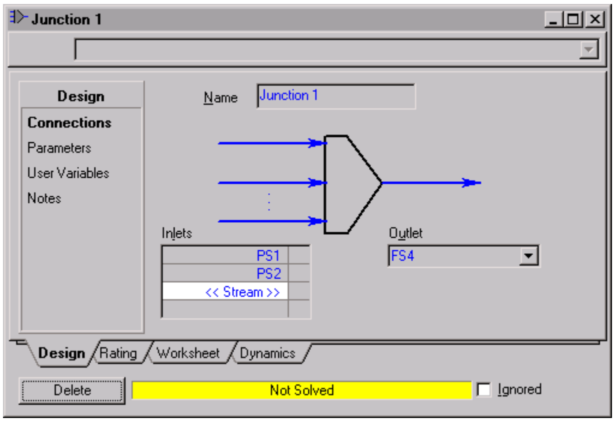
11.3.3 Adding a Mixer

For this pipeline configuration, a HYSYS **Mixer** is used to merge streams. Other HYSYS operations can be used to merge streams but the **Mixer** is the simplest to use and the most suitable for the this example.

*More information regarding HYSYS unite operations is located in the HYSYS Reference Manual 2, Chapter 13 - **Physical Operations**.*

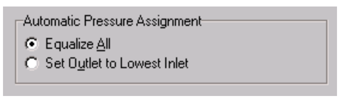
12. Add a **Mixer** to your simulation. Named **Junction 1**, it is used to merge streams **PS1** and **PS2**. Figure 11.9 shows the completed **Connections** page.

Figure 11.9



13. On the **Parameters** page of the Mixer, select the ***Equalize All*** radio button in the **Automatic Pressure Assignment** group box.

Figure 11.10



11.3.4 The Third PIPESYS Extension

The third PIPESYS extension you add is used to represent Branch 4. It consists of a single pipe unit.

14. Enter the data for the third PIPESYS extension as defined in the following table.

Connections Page	Name	Branch 4
	Inlet Stream	FS4
	Outlet Stream	PS4
	Energy Stream	ES4
Elevation Profile Page	Distance [ft]	0
	Elevation [ft]	2090
	Pipeline Unit	Pipe #1
	Pipe #1 Elevation [ft]	2077
	Pipe #1 Run [ft]	1167
Pipe Unit View	Nominal Diameter [Inches]	4
	Pipe Schedule	40
	Centre Line Depth	Default
Temperature Profile	Ambient Temperature [F]	40

11.3.5 The Fourth PIPESYS Extension

Branch 3 of this pipeline system is represented by the fourth PIPESYS extension. Three Pipe Units in the elevation profile matrix correctly characterize the changes in elevation occurring over the length of the pipeline.

To save time, add and define Pipe #1 and then use the Copy and Paste buttons to create Pipe #2 and Pipe #3.

15. The following table contains the information required to complete the fourth PIPESYS extension:

Connections Page	Name	Branch 3
	Inlet Stream	Well C
	Outlet Stream	PS3
	Energy Stream	ES3
Elevation Profile Page	Distance [ft]	0
	Elevation [ft]	2125
	Pipeline Unit	Pipe #1
	Pipe #1 Elevation [ft]	2077
	Pipe #1 Run [ft]	528
	Pipeline Unit	Pipe #2
	Pipe #2 Elevation [ft]	2080
	Pipe #2Run [ft]	334
	Pipeline Unit	Pipe #3
	Pipe #3 Elevation [ft]	2077
	Pipe #3 Run [ft]	670
Pipe Unit View (All Pipe Units Identical)	Nominal Diameter [Inches]	3
	Pipe Schedule	40
	Centre Line Depth	Default
Temperature Profile	Ambient Temperature [F]	40

11.3.6 The Fifth PIPESYS Extension

The fifth and final PIPESYS extension for this case represents Branch 5 of the pipeline system. In this segment, the total gas flows from Wells A, B and C are merged and the endpoint of the extension is the gas plant.

16. Enter the information for the final extension as defined in the following table:

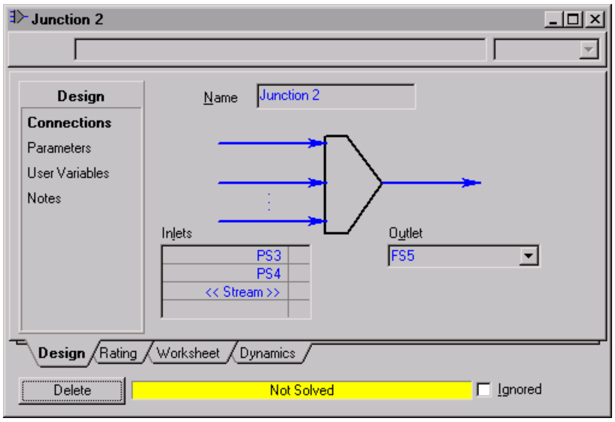
Connections Page	Name	Branch 5
	Inlet Stream	FS5
	Outlet Stream	PS5
	Energy Stream	ES5
Elevation Profile Page	Distance [ft]	0
	Elevation [ft]	2090
	Pipeline Unit	Pipe #1
	Pipe #1 Elevation [ft]	1980
	Pipe #1 Run [ft]	2110
Pipe Unit View	Nominal Diameter [Inches]	6
	Pipe Schedule	40
	Centre Line Depth	Default
Temperature Profile	Ambient Temperature [F]	40

11.3.7 The Second Mixer

A second Mixer merges the streams from Branches 3 and 4 with the outlet stream entering Branch 5.

17. Add a Mixer named **Junction 2** to your simulation. Inlet streams for the mixer are **PS3** and **PS4** and the outlet stream is **FS5**. See Figure 11.11 for the completed **Connections Page**:

Figure 11.11



18. On the **Parameters** page of the Mixer, select the ***Equalize All*** radio button in the **Automatic Pressure Assignment** group box.

11.3.8 Well Stream Information

To finish the case and have PIPESYS complete the calculation, the following stream parameters for the wells are required.

19. Finish specifying streams **Well A**, **Well B** and **Well C** with following data:

Well A	Temperature [F]	105
	Pressure [psia]	1060
	Molar Flow [MMSCFD]	8.6
Well B	Temperature [F]	115
	Molar Flow [MMSCFD]	7.4
Well C	Temperature [F]	110
	Molar Flow [MMSCFD]	10.1

20. Save your case as **network.hsc**.

11.4 Results

1. Go to the **Material Streams** page of the main Workbook. The results calculated for the product streams should appear as follows:

Figure 11.12

Name	Well A	Well B	Well C	PS1	PS2
Vapour Fraction	0.9655	0.9709	0.9688	0.9586	0.9640
Temperature [F]	105.0	115.0	110.0	91.81	100.4
Pressure [psia]	1060	1044	1018	993.3	993.3
Molar Flow [lbmole/hr]	944.3	812.6	1109	944.3	812.6
Mass Flow [lb/hr]	2.257e+004	1.942e+004	2.651e+004	2.257e+004	1.942e+004
Liquid Volume Flow [barrel/day]	4215	3627	4951	4215	3627
Heat Flow [Btu/hr]	-3.641e+007	-3.117e+007	-4.262e+007	-3.656e+007	-3.134e+007
Molar Enthalpy [Btu/lbmole]	-3.855e+004	-3.837e+004	-3.843e+004	-3.871e+004	-3.857e+004

Name	PS3	PS4	PS5	FS4	FS5
Vapour Fraction	0.9658	0.9597	0.9607	0.9612	0.9622
Temperature [F]	102.8	92.62	93.73	95.75	96.53
Pressure [psia]	972.1	972.1	963.0	993.3	972.1
Molar Flow [lbmole/hr]	1109	1757	2866	1757	2866
Mass Flow [lb/hr]	2.651e+004	4.199e+004	6.850e+004	4.199e+004	6.850e+004
Liquid Volume Flow [barrel/day]	4951	7842	1.279e+004	7842	1.279e+004
Heat Flow [Btu/hr]	-4.270e+007	-6.795e+007	-1.108e+008	-6.790e+007	-1.107e+008
Molar Enthalpy [Btu/lbmole]	-3.850e+004	-3.868e+004	-3.865e+004	-3.865e+004	-3.861e+004

To optimize the performance of the gas-condensate gathering system created in this example, see PIPESYS Application 2: ***Optimization Application.***

12 PIPESYS Application 2

12.1 Optimization Application 3

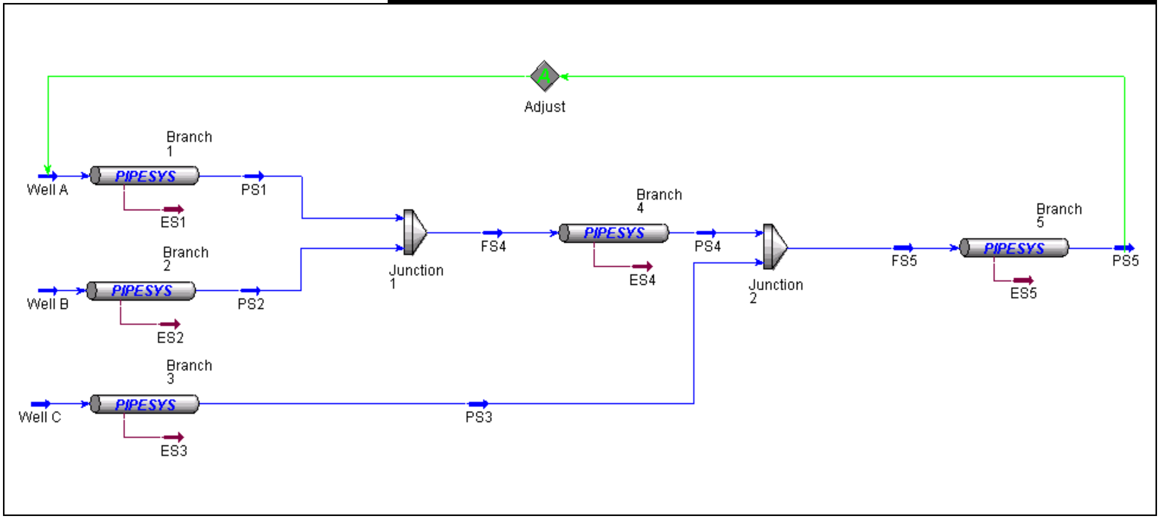
12.1 Optimization Application

Optimizing the Gas Condensate Gathering System

This application is a continuation of the **Application 1*: Gas-Condensate Gathering System***, in which you modelled the performance of a small gas-condensate gathering system given fixed wellhead rates and plant delivery requirements. As the next step, you will attempt to increase production from the wells by adding a compressor to the fifth PIPESYS extension. Using supplied wellhead performance curves, the effect of lowering the pressure at the wellheads will be gauged in terms of the resulting increased flow rates.

Figure 12.1 shows the completed PFD for the completed Application.

Figure 12.1



*You must complete the **Gas-Condensate Gathering System Application** before you are able to begin work on this application.*

1. Start HYSYS and load the case file ***network.hsc*** that you saved upon completion of the **Gas-Condensate Gathering System** of the first part of this application exercise.

The first modification you make will be to add an **ADJUST** operation. The adjust will be used to maintain a constant pressure of 1000 psia at the Gas Plant with the pressure at Well A being the adjusted variable.

2. Add an ADJUST operation with the following specifications.

Name	Adjust
Adjusted Variable - Object	Well A
Adjusted Variable - Variable	Pressure
Target Variable - Object	PS5
Target Variable - Variable	Pressure
Specified Target Variable	1000 psia
Method	Secant
Tolerance	0.10 psi
Step Size	100 psi
Max. Iter.	25

3. Press the ***Start*** button at the bottom of the Adjust property view to begin the adjust calculations. HYSYS will require several minutes to reach a solution. This is because the entire PIPESYS network must be recalculated for each iteration.

Upon convergence, the well pressures should be:

Well	Pressure (psia)
A	1093
B	1077
C	1052

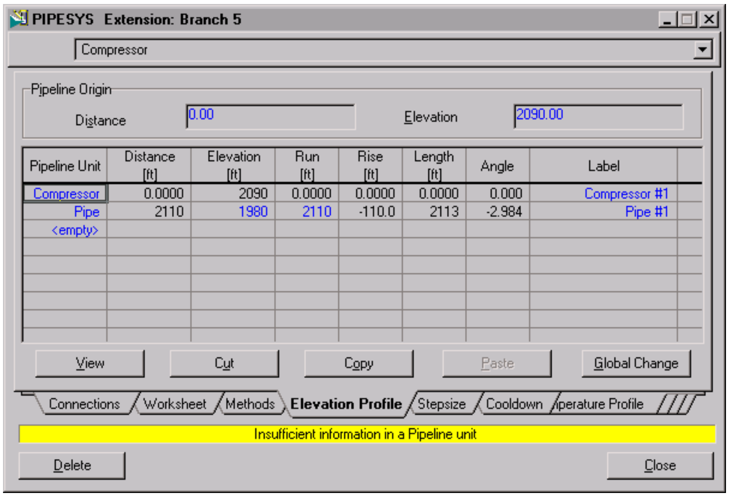
The next step will be to add an in-line compressor at the upstream end of PIPESYS Branch 5. If this addition reduces the pressure at the wells to an extent that production can be significantly increased, then the additional cost of the compressor is justifiable. It is also useful to look at the incremental performance increase of a larger compressor in order to get a feel for sizing the compressor. The performance of a 1000 hp compressor to that of a 750 hp compressor will be compared for this application.

*See the PIPESYS Manual, **Chapter 7 - In-line Compressor** for more information on adding and defining compressor parameters.*

4. Add a **Compressor** to the upstream end of the PIPESYS extension **Branch 5**. To do so, open the **Elevation Profile** page of Branch 5, click on the Pipe Unit and the select the **Compressor** from the Edit Bar drop-down list.

Figure 12.2 shows the **Elevation Profile Page** of Branch 5 with the added in-line compressor.

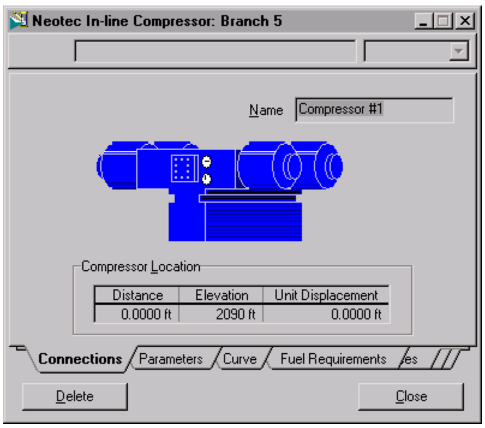
Figure 12.2



The performance of the 1000 hp compressor will be evaluated first.

Figure 2.3 shows the in-line compressor Connections page:

Figure 12.3



5. Use the following information to complete the Compressor Property view. On the **Parameters** page:

Brake Power - Specified	1000 hp
Max. Discharge Temp	100 F
Max. Interstage Temp	100 F
Number of Stages	2
Adiabatic Efficiency	0.73
Interstage delta P	10 psi

On the **Mechanical Losses** Page:

Overall Efficiency	0.95

Once again, the solution process will require several minutes to perform the iterative calculation for the PIPESYS network and converge.

When the process is complete, the well pressures should be as follows:

Well	Pressure (psia)
A	686.7
B	655.5
C	619.5

To compare the performance between two compressors, the same calculations are repeated using a **750 hp** compressor.

6. Change the **1000 hp** parameter in the **Specified** cell of the Brake Power group box to **750 hp**.

When the HYSYS completes the iteration, the new pressures are:

Well	Pressure (psia)
A	753.2
B	726.3
C	693.5

The PIPESYS calculations indicate that when a 1000 hp compressor is used the wellhead pressure is lower than when a 750 hp compressor is used. However, this may not result in an economically significant higher production rate, especially if these pressures are located on the steeper region of the wellhead performance curve. Figures 12.4, 12.5 and 12.6 at the end of this application show the wellhead performance curves for Well A, Well B and Well C, respectively. These curves can be used to evaluate compressor size that would be most economical for use in a particular pipeline network.

Locate 686.7 psia and 753.2 psia on the Well A wellhead curve and you should find that these correspond to flows of 11.1 MMSCFD and 10.8 MMSCFD respectively. This indicates that the 1000 hp compressor would increase production *by less than 5%*, over that of the 750 hp compressor. It is therefore reasonable to conclude that adding compression to the system is worthwhile since both compressors lower the wellhead pressures by a large amount, but the small increase in production may not be enough to justify the choice of the 1000 hp compressor. For this example, assume that economic and engineering considerations favour installing the 750 hp compressor.

In steps 4 through 6 it was determined that compression would significantly improve production and that the 750 hp compressor is the better candidate for doing so. Now you must find the actual flow rates and wellhead pressures that correspond to having the compressor in the system. This will be a process of adjusting the flow rates at each of the wells to manually converge on a particular point on the wellhead curves.

7. Locate the flow rates on the wellhead performance curves that correspond to the pressures calculated in Step 6. Reading from the curves these should be:

Well	Pressure (psia)	Flow (MMSCFD)
A	753.2	10.8
B	726.3	9.6
C	693.5	12.4

8. Press the ***Stop*** button and enter the flow rates from the above table into the stream data for the wells. Specify the a value of 10.8 MMSCFD for the Molar Flow of Well A. Similarly, enter the flow rates for Well B and Well C.
9. Press the ***Go*** button and let PIPESYS complete the iterations.

When the program is finished solving the network, the new well pressures calculated by PIPESYS should be:

Well	Pressure (psia)
A	868.5
B	838.0
C	783.0

10. Find the flow rates that correspond to these pressures from the wellhead curves. These values should be:

Well	Flow (MMSCFD)
A	10.3
B	9.0
C	11.9

11. Once again, press the ***Stop*** button and enter the flow rates from Step 11 into the well stream data as you did in Step 9.
12. Press the ***Go*** button. When the program is finished, the well pressures should read:

Well	Pressure (psia)
A	842.5
B	810.9
C	763.8

You will find that the flow rate and pressure for Wells B and C are close enough to the curves and can consider these to be a valid solution. However, the point 10.3 MMSCFD and 844.5 psia on the Well A Wellhead Performance curve is still some distance from the graph. You will need to do one or two more iterations to find the solution.

13. Find the flow rate on the Well A curve that corresponds to 844.5 psia, (this should be 10.5 MMSCFD). Press the ***Stop*** button and enter 10.5 into the flow rate parameter for the Well A stream.

14. Press the ***Go*** button. When the program is finished, pressures at the wells should be:

Well	Pressure (psia)
A	849.0
B	813.1
C	765.3

Now the pressure/flow rate for Well A is reasonably close to the curve. The pressures for B and C have changed a little but not significantly from the last iteration. Fortunately, the pressure at a given well is fairly insensitive to pressure changes at any of the other wells. This process can be repeated to obtain a solution of any arbitrary precision subject to the limits imposed by the computer, but this solution is accurate enough for further analysis.

Compression has increased flow rates by a considerable amount:

Well	Flow Without Compression (MMSCFD)	Flow With Compression (MMSCFD)
A	8.6	10.5
B	7.4	9.0
C	10.1	11.9

The engineering analysis shows that adding the compressor increased production by about 20% at each of the wells. These results can be used in an economic study to further examine the value of adding compression to the pipeline system.

Figure 12.4

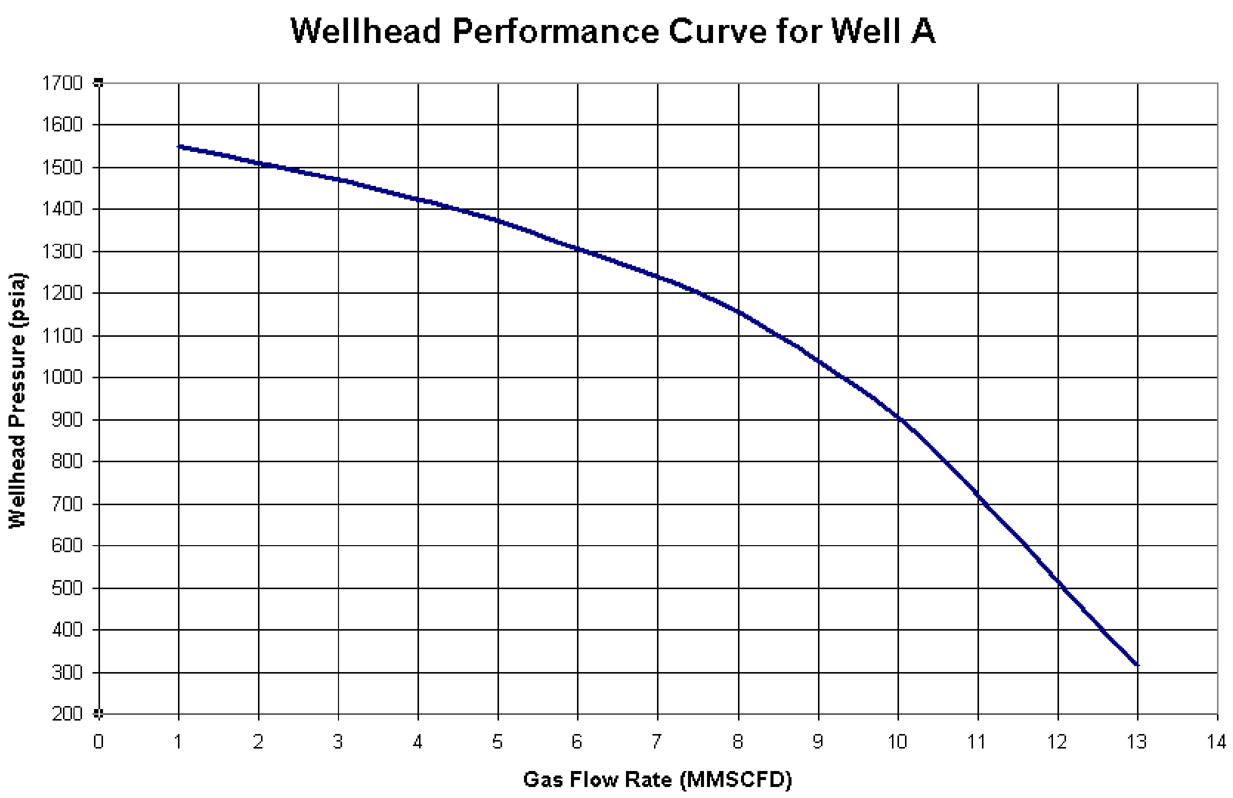

Wellhead Performance Curve for Well A
Wellhead Pressure (psia)
1700
1600
1500
1400
1300
1200
1100
1000
900
800
700
600
500
400
300
200
0
1
2
3
4
5
6
7
8
9
10
11
12
13
14
Gas Flow Rate (MMSCFD)

Figure 12.5

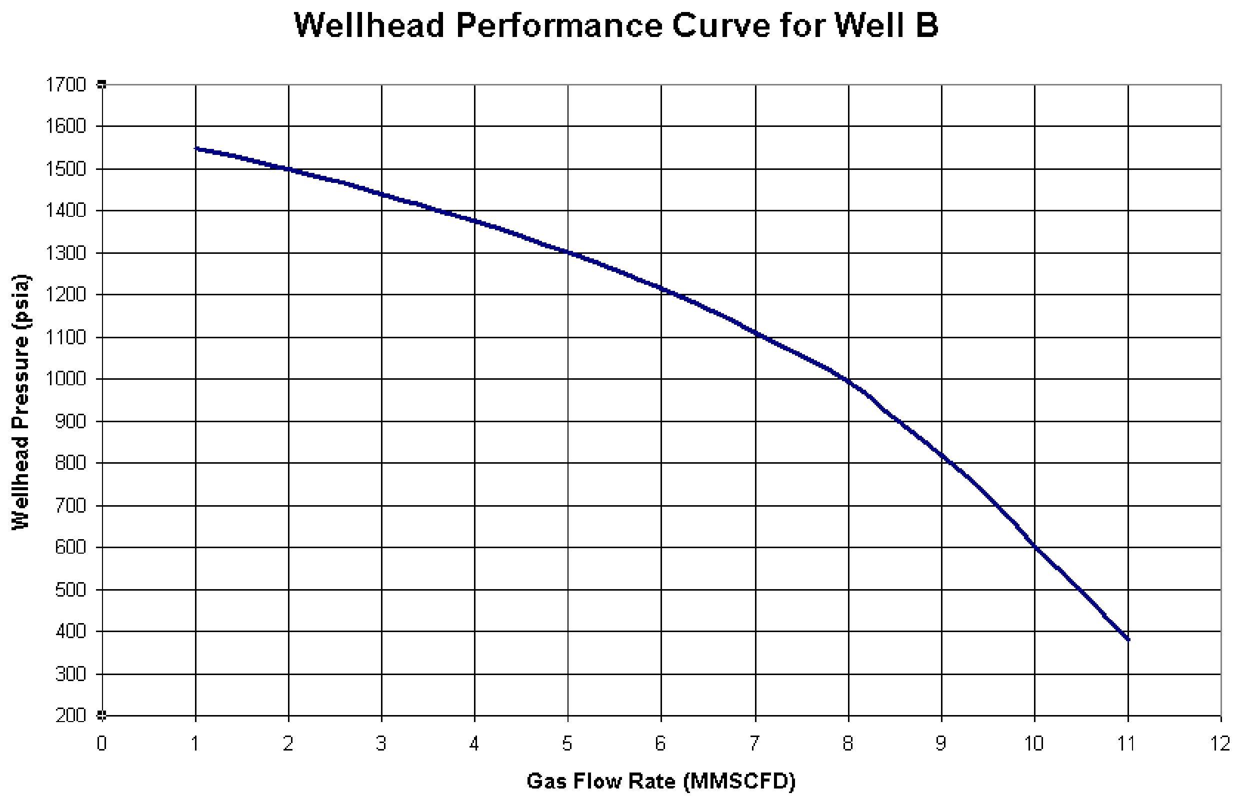
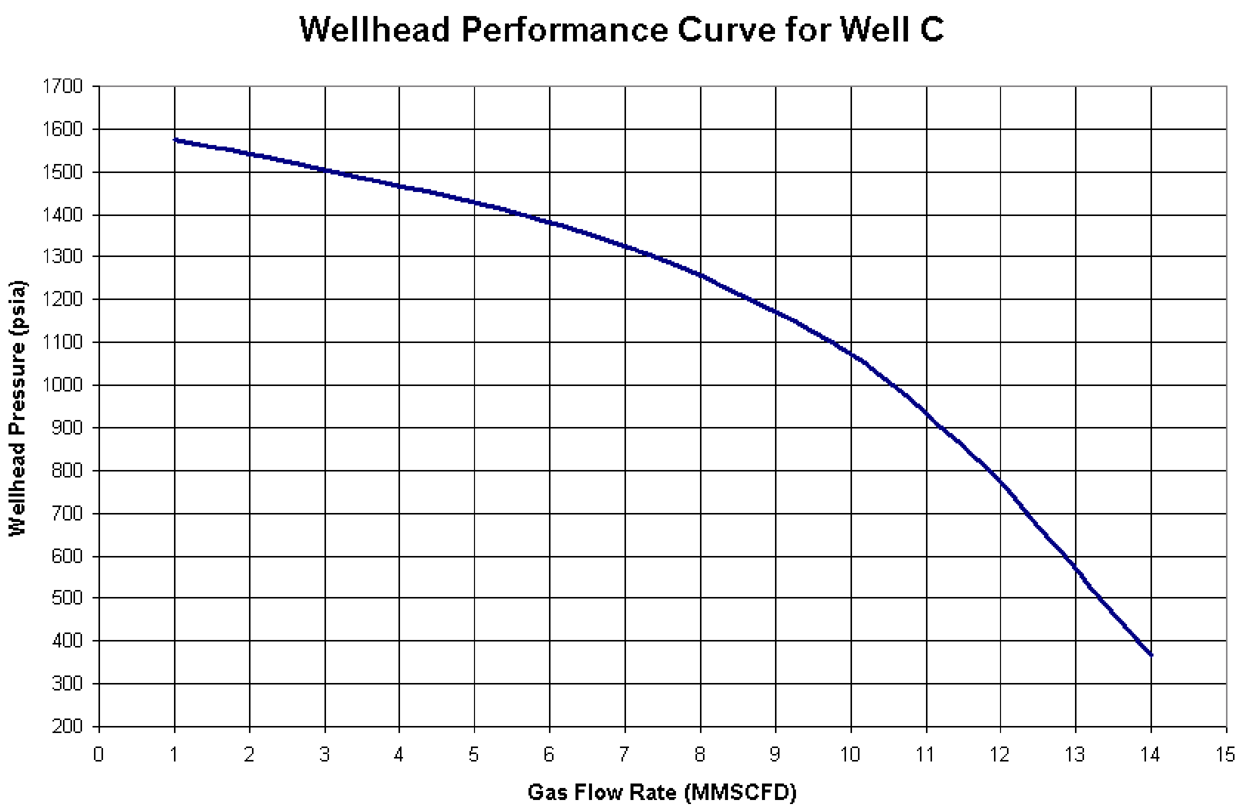

Wellhead Performance Curve for Well B
Wellhead Pressure (psia)
1700
1600
1500
1400
1300
1200
1100
1000
900
800
700
600
500
400
300
200
0
1
2
3
4
5
6
7
8
9
10
11
12
Gas Flow Rate (MMSCFD)

Figure 12.6


Wellhead Performance Curve for Well C
Wellhead Pressure (psia)
1700
1600
1500
1400
1300
1200
1100
1000
900
800
700
600
500
400
300
200
0
1
2
3
4
5
6
7
8
9
10
11
12
13
14
15
Gas Flow Rate (MMSCFD)

13 Glossary of Terms

13.1 PIPESYS Terms 3

13.2 References 6

13.3 PIPESYS Methods and Correlations 9

13.3.1 For Horizontal and Inclined Flow 9

13.3.2 For Vertical and Near Vertical Upflow and Downflow 10

13.1 PIPESYS Terms

absolute roughness: The effective roughness of a pipe, used in calculating the frictional pressure loss caused by fluid flow shear with the pipe wall. This quantity is assigned a value derived from a sand particle size such that a perfectly smooth pipe internally coated with sand particles of that size experiences a frictional pressure loss per unit length identical to that in the actual pipe, all other parameters being equal.

actual gas velocity: The velocity that is obtained when the in situ volumetric gas flow rate is divided by the cross sectional area of the pipe that is occupied by the gas

brake power: The power required to operate a compressor, including all losses experienced during gas compression and all mechanical losses.

component: All physical units that make up a pipeline. Has the same meaning as the term Pipeline Units.

elevation profile: A two-dimensional coordinate scheme for defining the path followed by a pipeline as it traverses its route. The elevation profile models the actual pipeline geometry as a series of connected straight-line segments, the end points of which are defined by horizontal and vertical displacement values.

gas power: The power required to operate a compressor, neglecting all mechanical losses.

in-line facility: Equipment used to handle or affect the fluid flow through a pipeline. Compressors, pumps, heaters and coolers are examples of in-line facilities.

inside film: The layer of fluid adjacent to the pipe inside wall (i.e. the boundary layer). The inside film is assumed to account for all resistance to heat transfer between the flowing fluid and the pipe wall.

inside film coefficient: A measure of the resistance to heat transfer between the fluid and the pipe wall due to convection effects.

Main PIPESYS View: The interface window to all of the user-definable characteristics of a PIPESYS extension. It is used to choose methods, add Pipeline Units, specify temperatures, and examine results.

mixture density: The density of a multiphase fluid mixture calculated as a volume weighted average of the gas and liquid densities.

mixture velocity: The average velocity of a multiphase fluid mixture calculated as the sum of the gas and liquid superficial velocities.

pig launcher: The point in a pipeline at which pigs are introduced into the fluid. stream.

pigging slug: The accumulation of liquid that builds up in front of a pig as it moves through a pipeline. In PIPESYS, the volume of the pigging slug is calculated to be the total initial volume of liquid in the pipeline (at steady state conditions), less the amount of liquid that flows out of the line during the transit time for the pig.

Pipe Unit: A straight-line segment of pipe connecting two points on an elevation profile. It is further defined by data such as diameter, roughness, heat transfer characteristics and environmental conditions.

Pipeline Origin: The location of the beginning of a pipeline. Vertical and horizontal coordinate values establish the physical location of the Pipeline Origin. These values are entered on the Elevation Profile page of the Main PIPESYS View, into the Distance and Elevation cells in the Pipeline Origin group box.

Pipeline Unit: This is an all-inclusive term that is used to refer to both Pipe Units and in-line facilities. All the physical units that make up a pipeline are referred to as Pipeline Units. This term can be used interchangeably with the term components.

relative roughness: The ratio of absolute roughness to the inside diameter of the pipe.

resistance coefficient: A dimensionless constant used to specify a pressure loss as a number of velocity heads.

step size: The initial length of pipe over which the pressure and/or temperature and/or enthalpy changes are computed for a Pipe Unit. If the computed change exceeds the maximum allowed in a step, then the length is shortened. When the optimizer option is selected the length of pipe for the calculation step will be increased if the computed change is less than the minimum specified. This length is further constrained by a minimum and maximum step size. If the software attempts calculation for a length of pipe smaller than the minimum step size, the calculations are terminated and a warning message issued.

superficial gas velocity: The volumetric gas rate divided by the total cross-sectional area of the pipe.

superficial liquid velocity: The volumetric liquid rate divided by the total cross-sectional area of the pipe.

theoretical power: The power required to operate a compressor assuming zero compression losses and an absence of mechanical losses.

velocity head: The portion of the total head of a fluid flow attributable to the velocity of the fluid. The velocity head is directly related to the kinetic energy component in the Bernoulli equation and is given by:

$$\frac{V^2}{\rho}$$

where: V = fluid velocity
ρ = fluid density

13.2 References

American Petroleum Institute, “Subsurface Controlled Subsurface Safety Valve Sizing Computer Program”,API Manual 14BM, Second Ed., p. 38, API, Jan. (1978)

American Petroleum Institute, Technical Data Book - Petroleum Refining, API, New York (1982)

Aziz, K., Govier, G.W., and Fogarasi, M.,“Pressure Drop in Wells Producing Oil and Gas”, J. Can. Petrol. Technol., Vol. 11, p. 38, July (1972)

Baker, O.“Simultaneous Flow of Oil and Gas”, Oil Gas J., Vol. 54, No. 12, p. 185, Jul. (1954)

Baker, O.“Experience with Two-Phase Pipelines”, Can. Oil Gas Ind., Vol. 14, No. 3, p. 43, Mar. (1961)

Beggs, H.D., and Brill, J.P.“A Study of Two-Phase Flow in Inclined Pipes”, J. Petrol. Technol., p. 607, May (1973)

Bendiksen, K.H., Maines, D., Moe, R., and Nuland, S.,“The Dynamic Two Fluid Model OLGA: Theory and Application”, SPE Paper No. 19451, SPE Prod. Eng., May (1991)

Burke, N.E., and Kashou, S.F.,“History Matching of a North Sea Flowline Startup Using OLGA Transient Multiphase Flow Simulator”, SPE Paper No. 24789, Presented at the 67th Annual SPE Technical Conference and Exhibition, Washington, DC, October (1992)

Chen, N.H., “An Explicit Equation for Friction Factor in Pipe”, Ind. Eng. Chem. Fund., Vol. 18, No. 3, p. 296 (1979)

Dukler, A.E., Wicks, M., and Cleveland, R., “Frictional Pressure Drop in Two-Phase Flow: B. An Approach Through Similarity Analysis”, AIChE J., Vol. 10, No. 1, p. 44, Jan. (1964)

Dukler, A.E., “Gas-Liquid Flow in Pipelines”, Monograph, Project NX-28, AGA/API, May (1969)

Duns, H., Jr., and Ros, N., “Vertical Flow of Gas and Liquid Mixtures in Wells”, Paper No. 22, Section II, World Petrol. Conf., Frankfurt, Germany (1963)

Eaton, B.A., Andrews, D.E., Knowles, C.R., Silberberg, I.H., and Brown, K.E.,"The Prediction of Flow Patterns, Liquid Holdup and Pressure Losses Occurring During Continuous Two-Phase Flow in Horizontal Pipelines", J. Petrol. Technol., p. 815, Jun. (1967)

Flanking, O., "Effect of Uphill Flow on Pressure Drop in Design of Two-Phase Gathering Systems", Oil Gas J., p. 132, Mar.10 (1958)

Fuchs, P.,"The Pressure Limit for Terrain Slugging", Paper B.4, Proc. of the 3rd Int. Conf. on Multiphase Flow, BHRA, The Hague, Netherlands (1987)

Govier, G.W., and Aziz, K.,The Flow of Complex Mixtures in Pipes, Van Nostrand-Reinhold, (1972), reprinted by Robert E. Krieger Publishing Co., Huntingdon, New York (1977)

Govier, G.W., and Fogarasi, M.,"Pressure Drop in Wells Producing Gas and Condensate", J. Can. Petrol. Technol., Oct. (1975)

Gregory, G.A.,"Estimation of the Overall Heat Transfer Coefficient for Calculating Heat Loss/Gain in Flowing Wells", Technical Note No. 4, Neotechnology Consultants Ltd., Calgary, Canada, Mar. (1991)

Gregory, G.A., Mandhane, J., and Aziz, K."Some Design Considerations for Two-Phase Flow in Pipes", J. Can. Petrol. Technol., Jan.-Mar. (1975)

Gregory, G.A.,"Comments on the Prediction of Minimum Unloading Velocities for Wet Gas Wells", Technical Note No. 14, Neotechnology Consultants Ltd., Calgary, Canada, Dec. (1989)

Gregory, G.A.,"Estimation of the Overall Heat Transfer Coefficient for the Calculation of Pipeline Heat Loss/Gain", Technical Note No.3, Neotechnology Consultants Ltd., Calgary, Canada, Oct. (1984), 1st Rev. Sept. (1990), 2nd Rev. Mar. (1991)

Hooper, W.B., "The Two-K Method Predicts Heat Losses in Pipe Fittings", Chem.Eng., p. 96, Aug. 24 (1981)

Hughmark, G.A., "Holdup and Heat Transfer in Horizontal Slug Gas-Liquid Flow", Chem. Eng. Sci., Vol 20, p. 1007 (1965)

Hughmark, G.A."Holdup in Gas-Liquid Flow", Chem. Eng. Prog., Vol. 58, No. 4, p. 62, Apr. (1962)

Lockhart, R.W., and Martinelli, R.C."Proposed Correlation of Data for Isothermal Two-Phase, Two-Component Flow in Pipes", Chem. Eng. Prog., Vol. 45, No. 1, p. 39, Jan. (1949)

Mandhane, J., Gregory, G., and Aziz, K., "A Flow Pattern Map for Gas-Liquid Flow in Horizontal Pipes", Int. J. Multiphase Flow, Vol. 1, p. 537 (1974)

Mandhane, J.M., Gregory, G.A., and Aziz, K., "Critical Evaluation of Friction Pressure-Drop Prediction Methods for Gas-Liquid Flow in Horizontal Pipes", J. Petrol. Technol., p. 1348, Oct. (1977)

Mukherjee, H., and Brill, J.P., "Liquid Holdup Correlations for Inclined Two-Phase Flow", J. Petrol. Technol., p. 1003, May (1983)

Oliemans, R.V.A., "Two-Phase Flow in Gas-Transmission Pipelines", Paper No. 76-Pet-25, Joint Petrol. Mech. Eng. & Pressure Vessels and Piping Conf., Mexico City, Mexico, Sept. (1976)

Oliemans, R.V.A., "Modelling of Gas-Condensate Flow in Horizontal and Inclined Pipes", Proc., ASME Pipeline Eng. Symp., ETCE, p. 73, Dallas, Texas, Feb. (1987)

Pots, B.F.M., Bromilow, I.G., and Konijn, M.J.W.F., "Severe Slug Flow in Offshore Flowline/Riser Systems", SPE Prod. Eng., p. 319, Nov. (1987)

Salama, M.M and Venkatesh, E.S., "Evaluation of API RP 14E Erosional Velocity Limitations for Offshore Gas Wells", Paper No. OTC 4485, presented at the 15th Annual Offshore Technology Conference, Houston, May (1983)

Singh, B., and Gregory, G.A., unpublished work (1983)

Taitel, Y., and Dukler, A. "A Model for Predicting Flow Regime Transitions in Horizontal and Near Horizontal Gas-Liquid Flow", AIChE J., Vol. 22, No. 1, p. 47, Jan. (1976)

Tennessee Gas Pipeline Co., private communication (1979)

13.3 PIPESYS Methods and Correlations

13.3.1 For Horizontal and Inclined Flow

Flow Regime Prediction Methods

- Beggs and Brill (1973)
- Beggs and Brill Revised (1977)
- Mandhane, Gregory and Aziz (1974)
- Mandhane, Gregory and Aziz Alternate (1974)
- Govier and Aziz (1972)
- Baker (1954)
- Taitel and Dukler (1976)
- OLGAS (1994)

Liquid Holdup Prediction Methods

- Olieman’s Mechanistic Model (1987)
- Hughmark (1962)
- Beggs and Brill (1973)
- Beggs and Brill Revised (1977)
- Dukler (1969)
- Eaton et al (1967)
- Lockhart and Martinelli (1949)
- OLGAS (1994)

Frictional Loss Prediction Methods

- Olieman’s Mechanistic Model (1987)
- Beggs and Brill (1973)
- Beggs and Brill Rough Pipe (1973)
- Olieman’s (1976)
- Lockhart and Martinelli (1949)
- Dukler et al (1964)
- Dukler et al Rough Pipe (1964)
- OLGAS (1994)

Uphill Corrections

- Beggs and Brill Liquid Holdup Correction
- Flanigan Head Correction Factor
- Tennessee Gas Head Factor
- Singh and Gregory Head Factor

13.3.2 For Vertical and Near Vertical Upflow and Downflow

Flow Regime Prediction Methods

- Govier and Aziz (1972)
- Beggs and Brill (1973)
- Beggs and Brill Revised (1977)
- OLGAS (1994)
- Gregory et al (1989)

Liquid Holdup Prediction Methods

- Aziz, Govier and Fogarasi (1972)
- Beggs and Brill (1973)
- Beggs and Brill Revised (1977)
- OLGAS (1994)
- Gregory et al (1989)

Frictional Loss Prediction Methods

- Aziz, Govier and Fogarasi (1972)
- Beggs and Brill (1973)
- Beggs and Brill Rough Pipe (1973)
- OLGAS (1994)
- Gregory et al (1989)

Index

A

Above Ground Pipe 5-5
Absolute Roughness 5-5
Activate Curve 8-5
Adiabatic Efficiency 7-6
Air Parameters 5-7
Angle 3-15
Attached Streams 3-10

B

Brake Power 7-5, 8-4
Buried Pipe 5-5
    Parameters 5-7

C

Calculate Temperature Profile 3-24
Calculation Methods
    Horizontal and Inclined Flow 13-9
    Vertical and Near Vertical Upflow and Downflow 13-10
Calculation Warnings 3-28
Compressor 7-1
    Actual Losses 7-11
    Connections Page 7-3
    Curve Page 7-7
    Exponent of Gas Power 7-11
    Manipulating Curve Data 7-8
    Mechanical Losses Page 7-10
    Optional Constraints 7-6
    Overall Efficiency 7-11
    Parameters 7-6
    Parameters Page 7-4
    Power Balance 7-10
    Requirements Page 7-11
    Results Page 7-12
Compressor. See In-line Compressor
Connections Page 3-10
Cooler 9-4
    Connections Page 9-5
    Parameters 9-5
    Parameters Page 9-5
Cut and Paste Functions 3-16

D

Default Methods
    Changing 3-12
Default Roughness 5-4
Disabling PIPESYS 3-10
Discharge Pressure 7-5
Distance 3-15

E

Elevation 3-15
Elevation Profile
    Defining 4-5
    Entering 3-15
    Parameters 3-15
Elevation Profile Page 3-13
Erosion Velocity Check 9-16
    Connections Page 9-17
    Limiting Velocity Equation 9-16, 9-17
    Max. Allowable Velocity 9-16
    Results Page 9-18

F

Features 3-3
Fittings 9-9
    Absolute Roughness 9-10
    Connections Page 9-9
    Inside Diameter 9-10
    Parameters Page 9-9
Fluid Systems 3-12
Fluid Temperature Options 3-13, 3-24
Force Enthalpy Convergence 3-20

G

Gas-Based Fluid Systems 3-12

Global Change 3-17, 6-1
  Example 6-9
  Procedure 6-7
Global Change View
  Connections Page 6-4
  Dimensions Page 6-5
  Heat Transfer Page 6-6
  Pipe Coatings 6-6
Glossary of Terms 13-1
Graphs
  Graph Control 3-28
  Plot Button 3-27
  Printing 3-27

H

Heat Capacity Ratio 7-7
Heat Transfer Environments 5-5
Heater 9-3
  Connections Page 9-3
  Parameters Page 9-3
Hooper K1 & Hooper K2 9-10

I

Ignoring PIPESYS 3-10
In-line Compressor
  Adding 7-12, 7-13
In-line Compressor. See Compressor.
In-line Pump. See Pump.
Inside Diameter 5-4
Inside Film Coefficient 5-6
Installation 2-1, 2-4
Interstage deltaP 7-7
Isentropic Compressor 7-5

L

Length 3-15
Liquid-Based Fluid Systems 3-12

M

Main PIPESYS View 2-6, 3-3, 3-9
Max Discharge Pressure 7-6
Max Discharge Temperature 7-6
Max Interstage Temperature 7-6
Max Power 7-6
Messages Page 3-28
Methods Page 3-11
Minimum Allowed Pressure 3-20

N

Nominal Diameter 5-4
Number of Stages 7-6

O

Outside Diameter 5-4

P

Pig Launcher 4-8, 9-12
Pigging Slug Size Check 9-11
  Connections Page 9-12
  Results Page 9-12
Pipe Coatings Page 5-8
Pipe Conductivity 5-6, 5-7
Pipe Schedule 5-4
Pipe Segment Results 3-25
Pipe Unit
  Adding 5-9
  Connections Page 5-3
  Dimensions 5-4
  Dimensions Page 5-3
  Global Change 6-3
  Heat Transfer Page 5-5
  View 5-1
Pipeline Origin 3-13
Pipeline Units 3-3, 3-14
PIPESYS and HYSYS
  Adding PIPESYS 3-4
  Property Views 3-9
PIPESYS Methods and Correlations 13-9
Plots. See Graphs.
Polytropic Compressor 7-5
Polytropic Efficiency 7-6
Pump 8-1
  Connections Page 8-3
  Curve Page 8-5
  Discharge Pressure 8-5
  Fluid Power 8-4
  Parameters Page 8-3
  Specified Discharge Pressure 8-5
Pump Efficiency 8-4

R

Rating Factor 7-7
References 13-6
Regulator 9-8
  Connections Page 9-8
  Parameters Page 9-8

Relative Roughness 5-5
Reports
    HYSYS Report Manager 3-26
    Neotec Maxi Report 3-27
    Neotec Mini Report 3-27
    Printing Reports 3-27
    Report Builder 3-26
Requirements
    Software 2-3
    System 2-3
Results Page 3-25
Rise 3-15
Run 3-15

S

Severe Slugging Check 9-13
    Connections Page 9-15
    Results Page 9-15
Soil Parameters 5-7
Stepsize Optimizer 3-19
Stepsize Page 3-18
Submerged Pipe 5-5
Suction Pressure 8-5

T

Technical Support 1-8
Temperature Profile Page 3-23

U

Unit-X 9-6
    Connections Page 9-6
    Parameters Page 9-7
User Interface 3-8

V

Valve Coefficient 9-10
Velocity Heads 9-10

W

Wall Thickness 5-4
Warranty 1-7
Water Parameters 5-7
Worksheet Page 3-11